FANUC Series 30*i*/300*i*/300*is*-MODEL A
FANUC Series 31*i*/310*i*/310*is*-MODEL A5
FANUC Series 31*i*/310*i*/310*is*-MODEL A
FANUC Series 32*i*/320*i*/320*is*-MODEL A

For Machining Center System

USER'S MANUAL

B-63944EN-2/02

In this manual we have tried as much as possible to describe all the various matters.
However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.
Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.

It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.

Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

CONTENTS

1.1 DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into **Warning** and **Caution** according to their bearing on safety. Also, supplementary information is described as a **Note**. Read the **Warning**, **Caution**, and **Note** thoroughly before attempting to use the machine.

> ⚠ **WARNING**
> Applied when there is a danger of the user being injured or when there is a danger of both the user being injured and the equipment being damaged if the approved procedure is not observed.

> ⚠ **CAUTION**
> Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

> **NOTE**
> The Note is used to indicate supplementary information other than Warning and Caution.

- Read this manual carefully, and store it in a safe place.

1.2 GENERAL WARNINGS AND CAUTIONS

⚠ **WARNING**

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
2. Before operating the machine, thoroughly check the entered data.
   Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate.
   If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
4. When using a tool compensation function, thoroughly check the direction and amount of compensation.
   Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

⚠ **WARNING**

5 The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6 Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

7 The User's Manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

8 Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.

⚠ **CAUTION**

The liquid-crystal display is manufactured with very precise fabrication technology. Some pixels may not be turned on or may remain on. This phenomenon is a common attribute of LCDs and is not a defect.

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off.
Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

1.3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied User's Manual carefully such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Coordinate system setting**
If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command. Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2 **Positioning by nonlinear interpolation**
When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming. Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Function involving a rotation axis**
When programming polar coordinate interpolation or normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely. Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4 **Inch/metric conversion**
Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

⚠ **WARNING**

5 **Constant surface speed control**
When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6 **Stroke check**
After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7 **Tool post interference check**
A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user. After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8 **Absolute/incremental mode**
If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9 **Plane selection**
If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10 **Torque limit skip**
Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

⚠ **WARNING**

11 **Programmable mirror image**
Note that programmed operations vary considerably when a programmable mirror image is enabled.

12 **Compensation function**
If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.
Before issuing any of the above commands, therefore, always cancel compensation function mode.

1.4 WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied User's Manual carefully, such that you are fully familiar with their contents.

⚠ **WARNING**

1 **Manual operation**
When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2 **Manual reference position return**
After switching on the power, perform manual reference position return as required.
If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed.
An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3 **Manual numeric command**
When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid.
Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4 **Manual handle feed**
In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

⚠ **WARNING**

5 **Disabled override**
If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6 **Origin/preset operation**
Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

7 **Workpiece coordinate system shift**
Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8 **Software operator's panel and menu switches**
Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9 **RESET key**
Pressing the RESET key stops the currently running program. As a result, the servo axes are stopped. However, the RESET key may fail to function for reasons such as an MDI panel problem. So, when the motors must be stopped, use the emergency stop button instead of the RESET key to ensure security.

⚠ **WARNING**

10 **Manual intervention**
If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

11 **Feed hold, override, and single block**
The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

12 **Dry run**
Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

13 **Cutter and tool nose radius compensation in MDI mode**
Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

14 **Program editing**
If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

1.5 WARNINGS RELATED TO DAILY MAINTENANCE

⚠ **WARNING**

1 **Memory backup battery replacement**
When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the Section “Method of replacing battery” in the User’s Manual (Common to T/M series) for details of the battery replacement procedure.

⚠ **WARNING**

2 **Absolute pulse coder battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

Refer to the FANUC SERVO MOTOR αi series Maintenance Manual for details of the battery replacement procedure.

⚠ **WARNING**

3 Fuse replacement

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

TABLE OF CONTENTS

III. OPERATION

APPENDIX

I. GENERAL

1 GENERAL

This manual consists of the following parts:

About this manual

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, characteristics, and restrictions.

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

APPENDIX
Lists parameters.

NOTE

1 This manual describes the functions that can operate in the machining center system path control type. For other functions not specific to the lathe system, refer to the User's Manual (Common to Lathe System/Machining Center System) (B-63944EN).

2 Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual (B-63942EN).

3 This manual does not detail the parameters not mentioned in the text. For details of those parameters, refer to the parameter manual (B-63950EN).
Parameters are used to set functions and operating conditions of a CNC machine tool, and frequently-used values in advance. Usually, the machine tool builder factory-sets parameters so that the user can use the machine tool easily.

4 This manual describes not only basic functions but also optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

Applicable models

The models covered by this manual, and their abbreviations are :

Model name	Abbreviation	
FANUC Series 30*i*-MODEL A	30*i* –A	Series 30*i*
FANUC Series 300*i*-MODEL A	300*i*–A	Series 300*i*
FANUC Series 300*is*-MODEL A	300*is*–A	Series 300*is*
FANUC Series 31*i*-MODEL A	31*i* –A	Series 31*i*
FANUC Series 31*i*-MODEL A5	31*i* –A5	
FANUC Series 310*i*-MODEL A	310*i*–A	Series 310*i*
FANUC Series 310*i*-MODEL A5	310*i*–A5	
FANUC Series 310*is*-MODEL A	310*is*–A	Series 310*is*
FANUC Series 310*is*-MODEL A5	310*is*–A5	
FANUC Series 32*i*-MODEL A	32*i* –A	Series 32*i*
FANUC Series 320*i*-MODEL A	320*i*–A	Series 320*i*
FANUC Series 320*is*-MODEL A	320*is*–A	Series 320*is*

NOTE

1. Unless otherwise noted, the model names 31*i*/310*i*/310*is*-A, 31*i*/310*i*/310*is*-A5, and 32*i*/320*i*/320*is*-A are collectively referred to as 30*i*/300*i*/300*is*. However, this convention is not necessarily observed when item 3 below is applicable.
2. Some functions described in this manual may not be applied to some products.
   For details, refer to the DESCRIPTIONS (B-63942EN).

Special symbols

This manual uses the following symbols:

- IP

Indicates a combination of axes such as X_ Y_ Z_
In the underlined position following each address, a numeric value such as a coordinate value is placed (used in PROGRAMMING.).

- ;

Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

Related manuals of
Series 30*i*/300*i*/300*i*s- MODEL A
Series 31*i*/310*i*/310*i*s- MODEL A
Series 31*i*/310*i*/310*i*s- MODEL A5
Series 32*i*/320*i*/320*i*s- MODEL A

The following table lists the manuals related to Series 30*i*/300*i* /300*i*s-A, Series 31*i*/310*i* /310*i*s-A, Series 31*i*/310*i* /310*i*s-A5, Series 32*i*/320*i* /320*i*s-A. This manual is indicated by an asterisk(*).

Table 1 Related manuals

Manual name	Specification number	
DESCRIPTIONS	B-63942EN	
CONNECTION MANUAL (HARDWARE)	B-63943EN	
CONNECTION MANUAL (FUNCTION)	B-63943EN-1	
USER'S MANUAL (Common to Lathe System/Machining Center System)	B-63944EN	
USER'S MANUAL (For Lathe System)	B-63944EN-1	
USER'S MANUAL (For Lathe Machining Center System)	B-63944EN-2	*
MAINTENANCE MANUAL	B-63945EN	
PARAMETER MANUAL	B-65950EN	
Programming		
Macro Compiler / Macro Executor PROGRAMMING MANUAL	B-63943EN-2	
Macro Compiler OPERATOR'S MANUAL	B-66264EN	
C Language Executor OPERATOR'S MANUAL	B-63944EN-3	
PMC		
PMC PROGRAMMING MANUAL	B-63983EN	
Network		
PROFIBUS-DP Board OPERATOR'S MANUAL	B-63994EN	
Fast Ethernet / Fast Data Server OPERATOR'S MANUAL	B-64014EN	
DeviceNet Board OPERATOR'S MANUAL	B-64044EN	
Operation guidance function		
MANUAL GUIDE *i* OPERATOR'S MANUAL	B-63874EN	
MANUAL GUIDE *i* Set-up Guidance OPERATOR'S MANUAL	B-63874EN−1	

Related manuals of SERVO MOTOR α*i*s/α*i*/β*i*s/β*i* series

The following table lists the manuals related to SERVO MOTOR α*i*s/α*i*/β*i*s/β*i* series

Table 2 Related manuals

Manual name	Specification number
FANUC AC SERVO MOTOR α*i*s series FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B-65262EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B-65272EN
FANUC AC SERVO MOTOR β*i*s series DESCRIPTIONS	B-65302EN
FANUC AC SPINDLE MOTOR β*i* series DESCRIPTIONS	B-65312EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B-65282EN
FANUC SERVO AMPLIFIER β*i* series DESCRIPTIONS	B-65322EN
FANUC SERVO MOTOR α*i*s series FANUC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series FANUC SERVO AMPLIFIER α*i* series MAINTENANCE MANUAL	B-65285EN
FANUC SERVO MOTOR β*i*s series FANUC AC SPINDLE MOTOR β*i* series FANUC SERVO AMPLIFIER β*i* series MAINTENANCE MANUAL	B-65325EN
FANUC AC SERVO MOTOR α*i*s series FANUC AC SERVO MOTOR α*i* series FANUC AC SERVO MOTOR β*i*s series PARAMETER MANUAL	B-65270EN
FANUC AC SPINDLE MOTOR α*i* series FANUC AC SPINDLE MOTOR β*i* series PARAMETER MANUAL	B-65280EN

Any of the servo motors and spindles listed above can be connected to the CNC described in this manual. However, α*i* series servo amplifiers can only be connected to α*i* series SVMs (for 30*i*/31*i*/32*i*).

This manual mainly assumes that the FANUC SERVO MOTOR α*i* series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

1.1 NOTES ON READING THIS MANUAL

⚠ **CAUTION**

1. The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.
2. In the header field of each page of this manual, a chapter title is indicated so that the reader can reference necessary information easily.
   By finding a desired title first, the reader can reference necessary parts only.
3. This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
   If a particular combination of operations is not described, it should not be attempted.

1.2 NOTES ON VARIOUS KINDS OF DATA

⚠ **CAUTION**

Machining programs, parameters, offset data, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.

II. PROGRAMMING

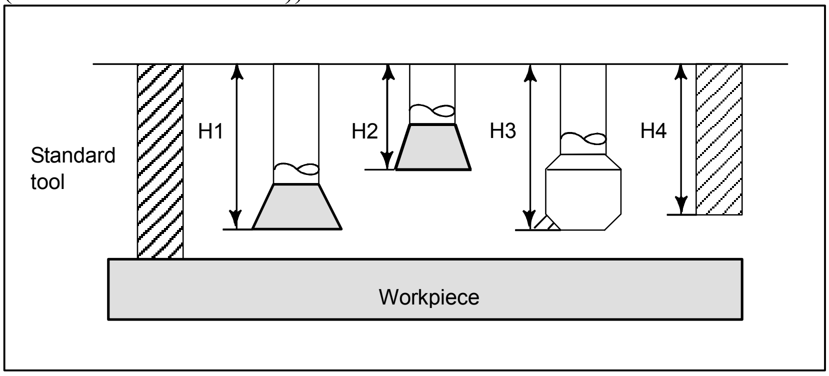
1 GENERAL

1.1 TOOL FIGURE AND TOOL MOTION BY PROGRAM

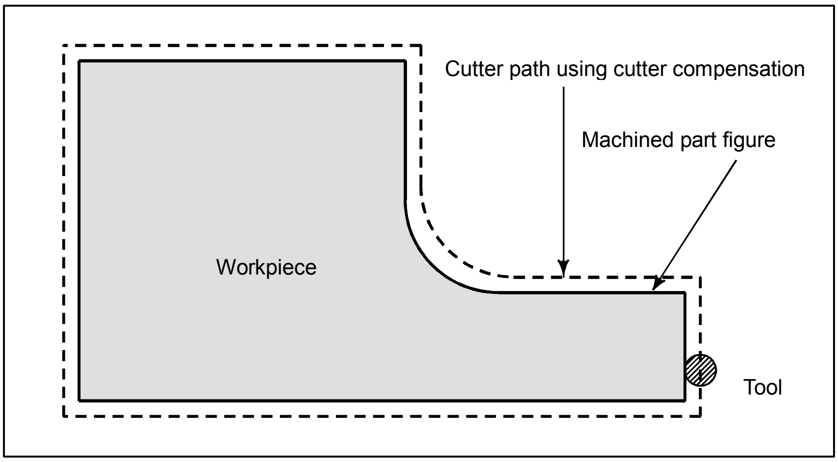
Explanation

- Machining using the end of cutter - Tool length compensation function

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.
Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (See Chapter "Setting and Displaying Data" in User's Manual (Common to T/M series)), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation (See Section "Tool Length Compensation" in User's Manual (Common to T/M series)).



- Machining using the side of cutter - Cutter compensation function



Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.
If radius of cutters are stored in the CNC (See Chapter "Setting and Displaying Data" in User's Manual (Common to T/M series)), the tool can be moved by cutter radius apart from the machining part figure. This function is called cutter compensation (See Section II-6 "Tool Compensation Function").

2 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One-shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)

G01 and G00 are modal G codes in group 01.

```
G01 X_ ;
    Z_ ;   G01 is effective in this range.
    X_ ;
G00 Z_ ;   G00 is effective in this range.
    X_ ;
G01 X_ ;
    :
```

Explanation

1. When the clear state (parameter CLR (No. 3402#6)) is set at power-up or reset, the modal G codes are placed in the states described below.
   (1) The modal G codes are placed in the states marked with ◤ as indicated in Table.
   (2) G20 and G21 remain unchanged when the clear state is set at power-up or reset.
   (3) Which status G22 or G23 at power on is set by parameter G23 (No. 3402#7). However, G22 and G23 remain unchanged when the clear state is set at reset.
   (4) The user can select G00 or G01 by setting parameter G01 (No. 3402#0).
   (5) The user can select G90 or G91 by setting parameter G91 (No. 3402#3).
   When G code system B or C is used in the lathe system, setting parameter G91 (No. 3402#3) determines which code, either G90 or G91, is effective.
   (6) In the machining center system, the user can select G17, G18, or G19 by setting parameters G18 and G19 (No. 3402#1 and #2).
2. G codes other than G10 and G11 are one-shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm PS0010 occurs.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle for drilling, the canned cycle for drilling is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle for drilling.
6. G codes are indicated by group.
7. The group of G60 is switched according to the setting of the parameter MDL (No. 5431#0). (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

Table 2(a) G code list

G code	Group	Function	
G00	01	Positioning (rapid traverse)	
G01		Linear interpolation (cutting feed)	
G02		Circular interpolation CW or helical interpolation CW	
G03		Circular interpolation CCW or helical interpolation CCW	
G02.2, G03.2		Involute interpolation CW/CCW	
G02.3, G03.3		Exponential interpolation CW/CCW	
G02.4, G03.4		Three-dimensional coordinate conversion CW/CCW	
G04	00	Dwell	
G05		AI contour control (high-precision contour control compatible command)	
G05.1		AI contour control / Nano smoothing / Smooth interpolation	
G05.4		HRV3,4 on/off	
G06.2	01	NURBS interpolation	
G07	00	Hypothetical axis interpolation	
G07.1 (G107)		Cylindrical interpolation	
G08		AI contour control (advanced preview control compatible command)	
G09		Exact stop	
G10		Programmable data input	
G10.6		Tool retract and recover	
G10.9		Programmable switching of diameter/radius specification	
G11		Programmable data input mode cancel	
G12.1	21	Polar coordinate interpolation mode	
G13.1		Polar coordinate interpolation cancel mode	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane selection	Xp: X axis or its parallel axis
G18		ZpXp plane selection	Yp: Y axis or its parallel axis
G19		YpZp plane selection	Zp: Z axis or its parallel axis
G20 (G70)	06	Input in inch	
G21 (G71)		Input in mm	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G25	19	Spindle speed fluctuation detection off	
G26		Spindle speed fluctuation detection on	
G27	00	Reference position return check	
G28		Automatic return to reference position	
G29		Movement from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.1		Floating reference position return	
G31		Skip function	
G31.8		EGB-axis skip	
G33	01	Threading	
G34		Variable lead threading	
G35		Circular threading CW	
G36		Circular threading CCW	
G37	00	Automatic tool length measurement	
G38		Cutter or tool nose radius compensation : preserve vector	
G39		Cutter or tool nose radius compensation : corner circular interpolation	

Table 2(a) G code list

G code	Group	Function
G40	07	Cutter or tool nose radius compensation : cancel Three-dimensional cutter compensation : cancel
G41		Cutter or tool nose radius compensation : left Three-dimensional cutter compensation : left
G41.2		Cutter compensation for 5-axis machining : left (type 1)
G41.3		Cutter compensation for 5-axis machining : (leading edge offset)
G41.4		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)
G41.5		Cutter compensation for 5-axis machining : left (type 1) (FS16i-compatible command)
G41.6		Cutter compensation for 5-axis machining : left (type 2)
G42		Cutter or tool nose radius compensation : right Three-dimensional cutter compensation : right
G42.2		Cutter compensation for 5-axis machining : right (type 1)
G42.4		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)
G42.5		Cutter compensation for 5-axis machining : right (type 1) (FS16i-compatible command)
G42.6		Cutter compensation for 5-axis machining : right (type 2)
G40.1	19	Normal direction control cancel mode
G41.1		Normal direction control on : right
G42.1		Normal direction control on : left
G43	08	Tool length compensation +
G44		Tool length compensation -
G43.1	08	Tool length compensation in tool axis direction
G43.4		Tool center point control (type 1)
G43.5		Tool center point control (type 2)
G45	00	Tool offset increase
G46		Tool offset decrease
G47		Tool offset double increase
G48		Tool offset double decrease
G49 (G49.1)	08	Tool length compensation cancel
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G50.2	31	Polygon turning cancel
G51.2		Polygon turning
G52	00	Local coordinate system setting
G53		Machine coordinate system setting
G53.1		Tool axis direction control
G54 (G54.1)	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63		Tapping mode
G64		Cutting mode
G65	00	Macro call

Table 2(a) G code list

G code	Group	Function
G66	12	Macro modal call A
G66.1		Macro modal call B
G67		Macro modal call A/B cancel
G68	16	Coordinate system rotation start or 3-dimensional coordinate conversion mode on
G69		Coordinate system rotation cancel or 3-dimensional coordinate conversion mode off
G68.2		Feature coordinate system selection
G72.1	00	Figure copy (rotation copy)
G72.2		Figure copy (linear copy)
G73	09	Peck drilling cycle
G74		Left-handed tapping cycle
G76		Fine boring cycle
G80		Canned cycle cancel
G80.5	24	Electronic gear box 2 pair: synchronization cancellation
G80.8	34	Electronic gear box: synchronization cancellation
G81	09	Drilling cycle or spot boring cycle
G81.1	00	Chopping
G81.5	24	Electronic gear box 2 pair: synchronization start
G81.8	34	Electronic gear box: synchronization start
G82	09	Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G84.2		Rigid tapping cycle (FS15 format)
G84.3		Left-handed rigid tapping cycle (FS15 format)
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute programming
G91		Incremental programming
G91.1	00	Checking the maximum incremental amount specified
G92		Setting for workpiece coordinate system or clamp at maximum spindle speed
G92.1		Workpiece coordinate system preset
G93	05	Inverse time feed
G94		Feed per minute
G95		Feed per revolution
G96	13	Constant surface speed control
G97		Constant surface speed control cancel
G98	10	Canned cycle : return to initial level
G99		Canned cycle : return to R point level
G107	00	Cylindrical interpolation
G112	21	Polar coordinate interpolation mode
G113		Polar coordinate interpolation mode cancel

3 INTERPOLATION FUNCTION

3.1 INVOLUTE INTERPOLATION (G02.2, G03.2)

Overview

Involute curve machining can be performed by using involute interpolation. Cutter compensation can be performed. Involute interpolation eliminates the need for approximating an involute curve with minute segments or arcs, and continuous pulse distribution is ensured even in high-speed operation of small blocks. Accordingly, high-speed operation can be performed smoothly. Moreover, machining programs can be created more easily, and the size of machining programs can be reduced.
In involute interpolation, the following two types of feedrate override functions are automatically executed, and a favorable cutting surface can be formed with high precision. (Automatic speed control function for involute interpolation)

- Override in cutter compensation mode
- Override in the vicinity of basic circle

Format

Involute interpolation on the Xp-Yp plane
G17 G02.2 Xp_ Yp_ I_ J_ R_ F_ ;
G17 G03.2 Xp_ Yp_ I_ J_ R_ F_ ;
Involute interpolation on the Zp-Xp plane
G18 G02.2 Zp_ Xp_ K_ I_ R_ F_ ;
G18 G03.2 Zp_ Xp_ K_ I_ R_ F_ ;
Involute interpolation on the Yp-Zp plane
G19 G02.2 Yp_ Zp_ J_ K_ R_ F_ ;
G19 G03.2 Yp_ Zp_ J_ K_ R_ F_ ;

Where,

G02.2	: Involute interpolation (clockwise)
G03.2	: Involute interpolation (counterclockwise)
G17/G18/G19	: Xp-Yp/Zp-Xp/Yp-Zp plane selection
Xp_	: X-axis or an axis parallel to the X-axis (specified in a parameter)
Yp_	: Y-axis or an axis parallel to the Y-axis (specified in a parameter)
Zp_	: Z-axis or an axis parallel to the Z-axis (specified in a parameter)
I_, J_, K_	: Center of the base circle for an involute curve viewed from the start point
R_	: Base circle radius
F_	: Cutting feedrate

Explanation

Involute curve machining can be performed by using involute interpolation. Involute interpolation ensures continuous pulse distribution even in high-speed operation in small blocks, thus enabling smooth and high-speed machining. Moreover, machining programs can be created more easily, and the size of machining programs can be reduced.

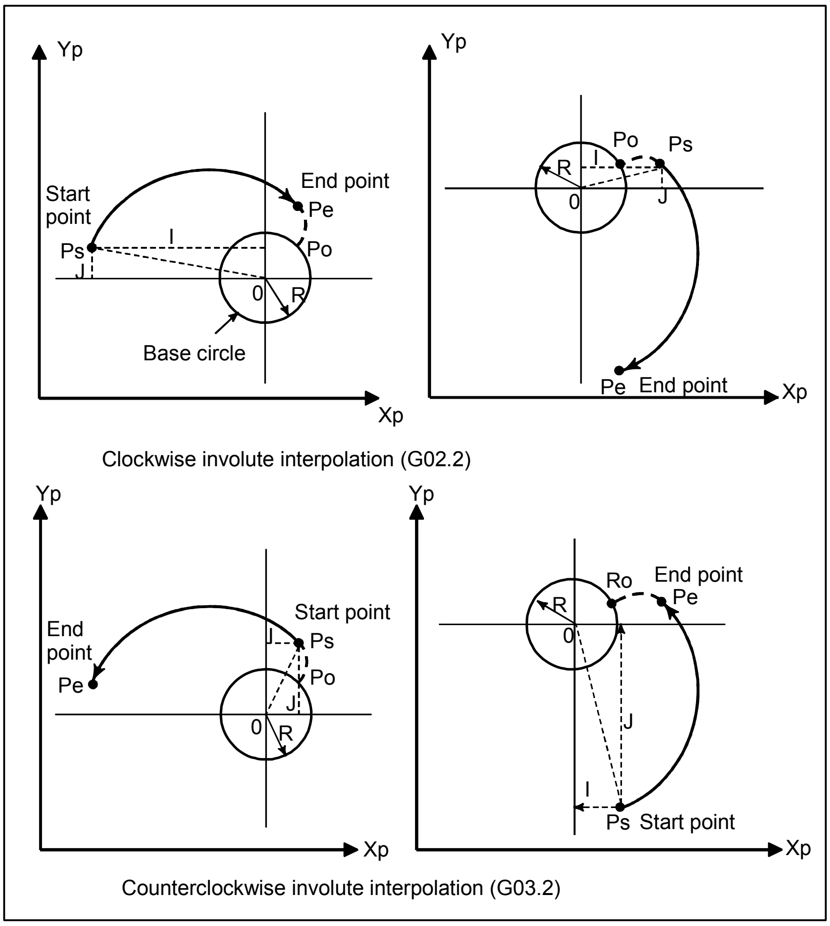


Fig. 3.1 (a) Actual movement

- Involute curve

An involute curve on the X-Y plane is defined as follows ;

$X(\theta) = R[\cos\theta + (\theta - \theta_O)\sin\theta] + X_O$

$Y(\theta) = R[\sin\theta - (\theta - \theta_O)\cos\theta] + Y_O$

where,

X_O, Y_O	:	Coordinates of the center of a base circle
R	:	Base circle radius
θ_O	:	Angle of the start point of an involute curve
θ	:	Angle of the point where a tangent from the current position to the base circle contacts the base circle
$X(\theta)$, $Y(\theta)$	:	Current position on the X-axis and Y-axis

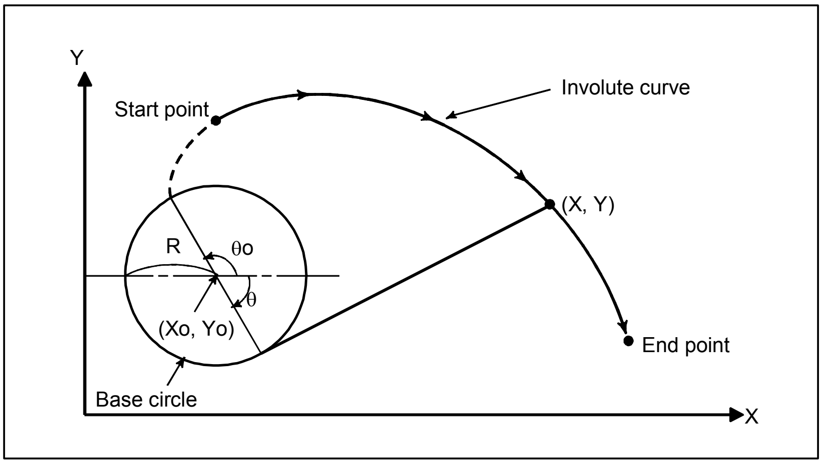


Fig. 3.1 (b) Involute curve

Involute curves on the Z-X plane and Y-Z plane are defined in the same way as an involute curve on the X-Y plane.

- Start point and end point

The end point of an involute curve is specified using address Xp, Yp, or Zp. An absolute value or incremental value is used to specify an Xp, Yp, or Zp value. When using an incremental value, specify the coordinates of the end point viewed from the start point of the involute curve.
When no end point is specified, alarm PS0241 is issued.
If the specified start point or end point lies within the base circle, alarm PS0242 is issued. The same alarm is issued if cutter compensation C causes the offset vector to enter the base circle. Be particularly careful when applying an offset to the inside of an involute curve.

- Base circle specification

The center of a base circle is specified with I, J, and K, corresponding to X, Y, and Z. The value following I, J, or K is a vector component defined when the center of the base circle is viewed from the start point of the involute curve; this value must always be specified as an incremental value, regardless of the G90/G91 setting. Assign a sign to I, J, and K according to the direction.
If I, J, and K are all left unspecified, or I0, J0, K0 is specified, alarm PS0241 or PS0242 is issued.
If R is not specified, or $R \leq 0$, alarm PS0241 or PS0242 is issued.

- Choosing from two types of involute curves

When only a start point and I, J, and K data are given, two types of involute curves can be created. One type of involute curve extends towards the base circle, and the other extends away from the base circle. When the specified end point is closer to the center of the base circle than the start point, the involute curve extends toward the base circle. In the opposite case, the involute curve extends away from the base circle.

- Feedrate

The cutting feedrate specified in an F code is used as the feedrate for involute interpolation. The feedrate along the involute curve (feedrate along the tangent to the involute curve) is controlled to satisfy the specified feedrate.

- Plane selection

As with circular interpolation, the plane to which to apply involute interpolation can be selected using G17, G18, and G19.

- Cutter compensation

Cutter compensation can be applied to involute curve machining. As with linear and circular interpolation, G40, G41, and G42 are used to specify cutter compensation.

G40: Cutter compensation cancel
G41: Cutter compensation left
G42: Cutter compensation right

First, a point of intersection with a segment or an arc is approximated both at the start point and at the end point of the involute curve. An involute curve passing the two approximated points of intersection with the start point and end pint becomes the tool center path.
Before selecting the involute interpolation mode, specify G41 or G42, cancel involute interpolation, and then specify G40. G41, G42, and G40 for cutter compensation cannot be specified in the involute interpolation mode.

- Automatic speed control

Cutting precision can be improved by automatically overriding the programmed feedrate during involute interpolation. See a subsequent subsection, "Automatic Speed Control for Involute Interpolation."

- Specifiable G codes

The following G codes can be specified in involute interpolation mode:

G04: Dwell
G10: Programmable data input
G17: X-Y plane selection
G18: Z-X plane selection
G19: Y-Z plane selection
G65: Macro call
G66: Macro modal call
G67: Macro modal call cancel
G90: Absolute programming
G91: Incremental programming

- Modes that allow involute interpolation specification

Involute interpolation can be specified in the following G code modes:

G41 : Cutter compensation left
G42 : Cutter compensation right
G51 : Scaling
G51.1 : Programmable mirror image
G68 : Coordinate rotation

- End point error

As shown below the end point may not be located on an involute curve that passes through the start point.
When an involute curve that passes through the start point deviates from the involute curve that passes through the end point by more than the value set in parameter No. 5610, alarm PS0243 is issued.
If there is an end point error, the programmed feedrate changes by the amount of error.

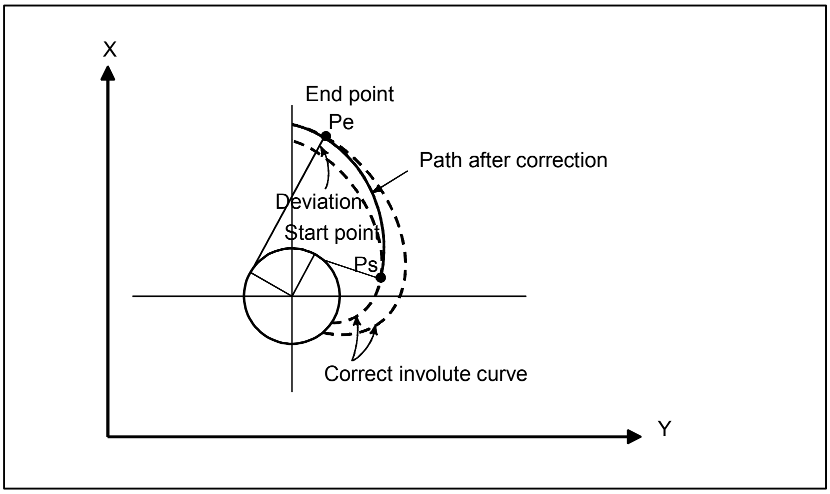


Fig. 3.1 (c) End point error in counterclockwise involute interpolation (G03.2)

3.1.1 Automatic Speed Control for Involute Interpolation

This function automatically overrides the programmed feedrate in two different ways during involute interpolation. With this function, a favorable cutting surface can be formed with high precision.

- Override in cutter compensation mode
- Override in the vicinity of basic circle

- Override in cutter compensation mode

When cutter compensation is applied to involute interpolation, control is exercised in ordinary involute interpolation so that the tangential feedrate on the tool-center path always keeps the specified feedrate.
Under the control, the actual cutting feedrate (feedrate around the perimeter of the tool (cutting point) on the path specified in the program) changes because the curvature of the involute curve changes every moment.
If the tool is offset in the inward direction of the involute curve in particular, the actual cutting feedrate becomes higher than the specified feedrate as the tool gets nearer to the base circle.
For smooth machining, it is desirable to control the actual cutting feedrate so that the feedrate keeps the specified feedrate. This function calculates an appropriate override value for the ever-changing curvature of the involute curve in the involute interpolation mode after cutter compensation. The function also controls the actual cutting feedrate (tangential feedrate at the cutting point) so that it always keeps the specified feedrate.

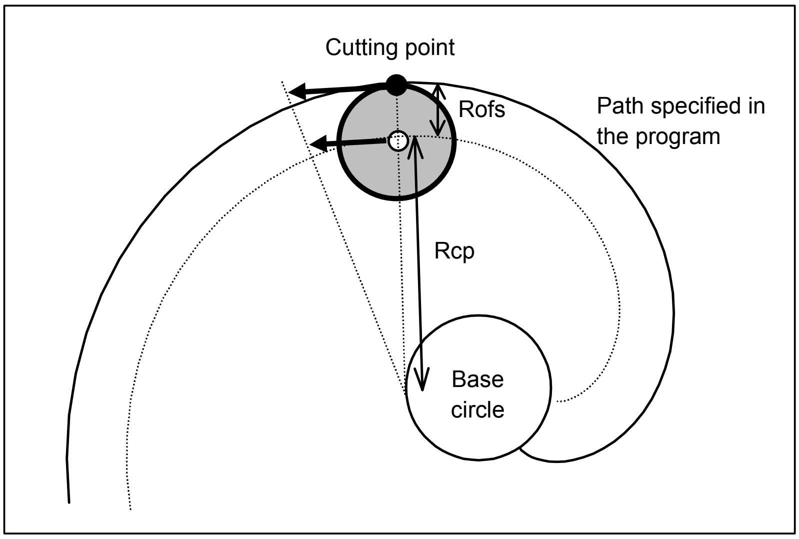


Fig. 3.1 (d) Override for inward offset by cutter compensation

Inward offset

$$OVR = Rcp/(Rcp + Rofs) \times 100$$

Outward offset

$$OVR = Rcp/(Rcp - Rofs) \times 100$$

where,

Rcp : Radius of curvature at the center of the tool of the involute curve passing through the center of the tool

Rofs : Radius of the cutter

- Clamping the override

The lower limit of override is specified in parameter No. 5620 so that the override for inward offset by cutter compensation or the override in the vicinity of the basic circle will not bring the speed of the tool center to zero in the vicinity of the basic circle.
The lower limit of override (OVR1o) is specified in parameter No. 5620 so that the inward offset will not reduce the speed of the tool center to a very low level in the vicinity of the basic circle.
Accordingly, the feedrate is clamped but does not fall below the level determined by the programmed feedrate and the lower limit of override (OVR1o).
The outward offset may increase the override to a very high level, but the feedrate will not exceed the maximum cutting feedrate.

- Clamping the acceleration in the vicinity of basic circle

If the acceleration calculated from the radius of curvature of the involute curve exceeds a value specified in the corresponding parameter, the tangential velocity is controlled so that the actual acceleration will not exceed the value specified in the parameter. Because the acceleration is always limited to a constant level, efficient velocity control can be performed for each machine. Because smooth velocity control can be performed continuously, impacts in machining in the vicinity of the basic circle can be reduced.
To calculate the acceleration, the radius of curvature of the involute curve and the tangential velocity are substituted into the following formula of circular acceleration:

$$\text{Acceleration} = F \times F/R$$

F: Tangential velocity

R: Radius of curvature

The maximum permissible acceleration is specified in parameter No. 1735.
If the calculated acceleration exceeds the maximum permissible acceleration, the feedrate is clamped to the level calculated by the following expression:

$$\text{Clamp level} = \sqrt{\text{Radius of curvature} \times \text{Maximum permissible acceleration}}$$

If the calculated clamp level falls below the lower limit of feedrate, the lower limit of feedrate becomes the clamp level. The lower limit of feedrate is specified in parameter No. 1732.

3.1.2 Helical Involute Interpolation (G02.2, G03.2)

As with arc helical involute interpolation, this function performs helical involute interpolation on the two axes involute interpolation and on up to four other axes simultaneously.

Format

Helical involute interpolation in Xp-Yp plane

$$\text{G17}\ \begin{Bmatrix}\text{G02.2}\\ \text{G03.2}\end{Bmatrix}\ \text{Xp\_Yp\_I\_J\_R\_}\alpha\text{\_}\beta\text{\_}\gamma\text{\_}\delta\text{\_F\_;}$$

Helical involute interpolation in Zp-Xp plane

$$\text{G18}\ \begin{Bmatrix}\text{G02.2}\\ \text{G03.2}\end{Bmatrix}\ \text{Zp\_Xp\_K\_I\_R\_}\alpha\text{\_}\beta\text{\_}\gamma\text{\_}\delta\text{\_F\_;}$$

Helical involute interpolation in Yp-Zp plane

$$\text{G19}\ \begin{Bmatrix}\text{G02.2}\\ \text{G03.2}\end{Bmatrix}\ \text{Yp\_Zp\_J\_K\_R\_}\alpha\text{\_}\beta\text{\_}\gamma\text{\_}\delta\text{\_F\_;}$$

$\alpha, \beta, \gamma, \delta$: Optional axis other than the axes of involute interpolation. Up to four axes can be specified.

3.1.3 Involute Interpolation on Linear Axis and Rotary Axis (G02.2, G03.2)

By performing involute interpolation in the polar coordinate interpolation mode, involute cutting can be carried out. Cutting is performed along an involute curve drawn in the plane formed by a linear axis and a rotary axis.

Format

If the linear axis is the X-axis or an axis parallel to the X-axis, the plane is considered to be the Xp-Yp plane, and I and J are used.

$$\left\{\begin{array}{l}\mathbf{G02.2}\\ \mathbf{G03.2}\end{array}\right\}\ \mathbf{X\_\_C\_\_I\_\_J\_\_R\_\_F\_\_;}$$

If the linear axis is the Y-axis or an axis parallel to the Y-axis, the plane is considered to be the Yp-Zp plane, and J and K are used.

$$\left\{\begin{array}{l}\mathbf{G02.2}\\ \mathbf{G03.2}\end{array}\right\}\ \mathbf{Y\_\_C\_\_J\_\_K\_\_R\_\_F\_\_;}$$

If the linear axis is the Z-axis or an axis parallel to the Z-axis, the plane is considered to be the Zp-Xp plane, and K and I are used.

$$\left\{\begin{array}{l}\mathbf{G02.2}\\ \mathbf{G03.2}\end{array}\right\}\ \mathbf{Z\_\_C\_\_K\_\_I\_\_R\_\_F\_\_;}$$

G02.2 : Clockwise involute interpolation
G03.2 : Counterclockwise involute interpolation
Example) If the linear axis is the X-axis
X, C : End point of the involute curve
I, J : Center of the basic circle of the involute curve, viewed from the start point
R : Radius of basic circle
F : Cutting feedrate

Example

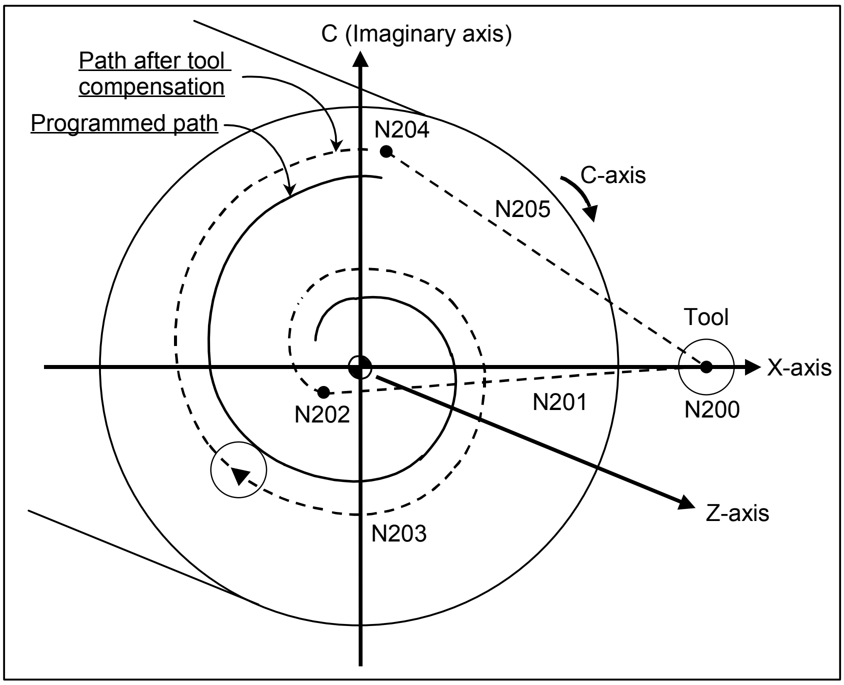


Fig. 3.1 (e) Involute interpolation in the polar coordinate interpolation mode

```
O0001 ;
   .
   .
N010 T0101 ;
   .
   .
N100 G90 G00 X15.0 C0 Z0 ;               Positioning to the start point
N200 G12.1 ;                             Polar coordinate interpolation
N201 G41 G00 X-1.0 ;                     started
N202 G01 Z-2.0 F__ ;
N203 G02.2 X1.0 C9.425 I1.0 J0 R1.0 ;    Involute interpolation during polar
N204 G01 Z0 ;                            coordinate interpolation
N205 G40 G00 X15.0 C0 ;
N206 G13.1 ;                             Polar coordinate interpolation
N300 Z__ ;                               cancelled
N400 X__ C__ ;
   .
   .
M30 ;
```

Limitation

- Number of involute curve turns

Both the start point and end point must be within 100 turns from the point where the involute curve starts. An involute curve can be specified to make one or more turns in a single block.
If the specified start point or end point is beyond 100 turns from the point where the involute curve starts, alarm PS0242 is issued.

- Unspecifiable functions

In involute interpolation mode, optional chamfering and corner R cannot be specified.

- Mode that does not allow involute interpolation specification

Involute interpolation cannot be used in the following mode:
G07.1: Cylindrical interpolation

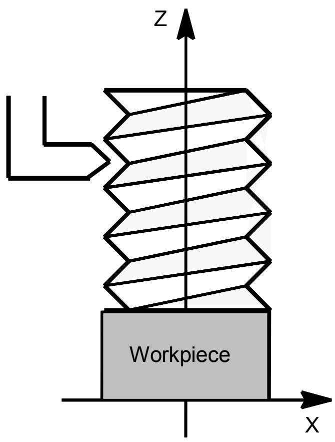
3.2 THREADING (G33)

Straight threads with a constant lead can be cut. The position coder mounted on the spindle reads the spindle speed in real-time. The read spindle speed is converted to the feedrate per minute to feed the tool.

Format

G33IP_ F_ ;

F : Long axis direction lead



Explanation

In general, threading is repeated along the same tool path in rough cutting through finish cutting for a screw. Since threading starts when the position coder mounted on the spindle outputs a 1-turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated threading. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.
Table 3.2 (a) lists the ranges for specifying the thread lead.

Table 3.2 (a) Ranges of lead sizes that can be specified

	Least command increment	Command value range of the lead
Metric input	0.001 mm	F1 to F50000 (0.01 to 500.00mm)
	0.0001 mm	F1 to F50000 (0.01 to 500.00mm)
Inch input	0.0001 inch	F1 to F99999 (0.0001 to 9.9999inch)
	0.00001 inch	F1 to F99999 (0.0001 to 9.9999inch)

> **NOTE**
> 1 The spindle speed is limited as follows :
> $1 \leq$ spindle speed $\leq$ (Maximum feedrate) / (Thread lead)
> Spindle speed : min^{-1}
> Thread lead : mm or inch
> Maximum feedrate : mm/min or inch/min ; maximum command-specified feedrate for feed-per-minute mode or maximum feedrate that is determined based on mechanical restrictions including those related to motors, whichever is smaller
> 2 Cutting feedrate override is not applied to the converted feedrate in all machining process from rough cutting to finish cutting. The feedrate is fixed at 100%
> 3 The converted feedrate is limited by the upper feedrate specified.
> 4 Feed hold is disabled during threading. Pressing the feed hold key during threading causes the machine to stop at the end point of the next block after threading (that is, after the G33 mode is terminated)

Example

Threading at a pitch of 1.5mm

G33 Z10. F1.5;

4 COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

4.1 POLAR COORDINATE COMMAND (G15, G16)

The end point coordinate value can be input in polar coordinates (radius and angle).
The plus direction of the angle is counterclockwise of the selected plane first axis + direction, and the minus direction is clockwise.
Both radius and angle can be commanded in either absolute or incremental programming (G90, G91).

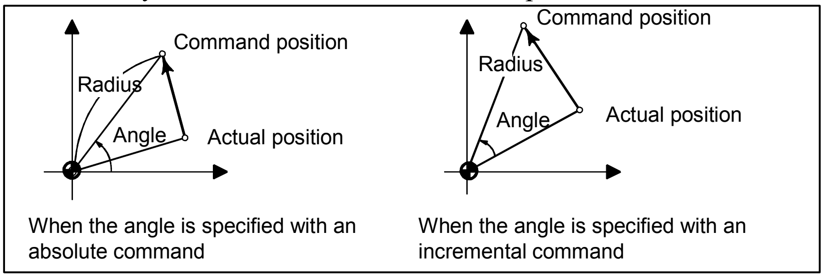
Format

Gxx Gyy G16;	**Starting the polar coordinate command (polar coordinate mode)**
G00 IP_ ; **:** **:**	**Polar coordinate command**
G15;	**Canceling the polar coordinate command (polar coordinate mode)**

G16 : Polar coordinate command
G15 : Polar coordinate command cancel
Gxx : Plane selection of the polar coordinate command (G17, G18 or G19)
Gyy : Center selection of the polar coordinate command (G90 or G91)
G90 specifies the origin of the workpiece coordinate system as the origin of the polar coordinate system, from which a radius is measured.
G91 specifies the current position as the origin of the polar coordinate system, from which a radius is measured.
IP_ : Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their values
First axis : radius of polar coordinate
Second axis : angle of polar coordinate

- Setting the origin of the workpiece coordinate system as the origin of the polar coordinate system

Specify the radius (the distance between the origin and the point) to be programmed with an absolute programming. The origin of the workpiece coordinate system is set as the origin of the polar coordinate system.
When a local coordinate system (G52) is used, the origin of the local coordinate system becomes the center of the polar coordinates.

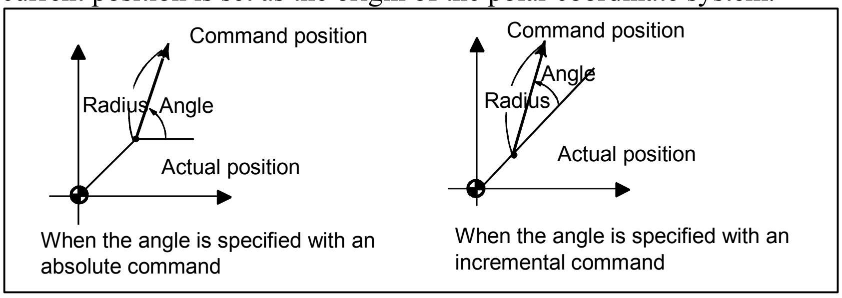


When the angle is specified with an absolute command

When the angle is specified with an incremental command

- Setting the current position as the origin of the polar coordinate system

Specify the radius (the distance between the current position and the point) to be programmed with an incremental programming. The current position is set as the origin of the polar coordinate system.

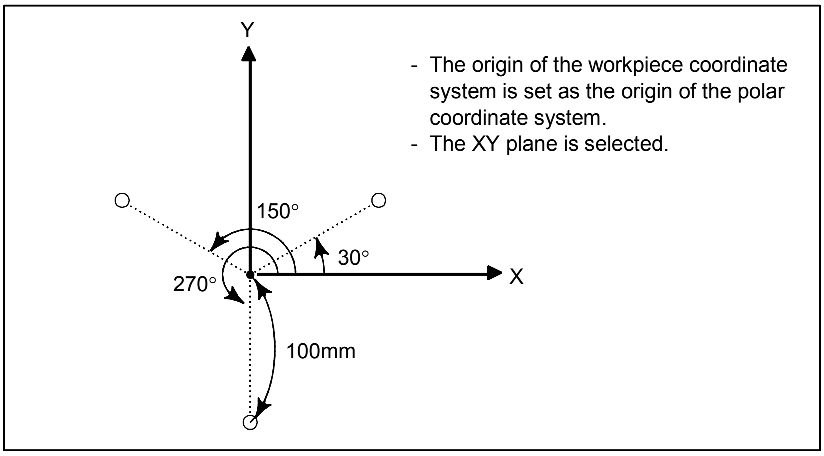


When the angle is specified with an absolute command

When the angle is specified with an incremental command

Example

Bolt hole circle



- Specifying angles and a radius with absolute programmings

N1 G17 G90 G16 ;	Specifying the polar coordinate command and selecting the XY plane Setting the origin of the workpiece coordinate system as the origin of the polar coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;	
	Specifying a distance of 100 mm and an angle of 30 degrees
N3 Y150.0 ;	Specifying a distance of 100 mm and an angle of 150 degrees
N4 Y270.0 ;	Specifying a distance of 100 mm and an angle of 270 degrees
N5 G15 G80 ;	Canceling the polar coordinate command

- Specifying angles with incremental programmings and a radius with absolute programmings

N1 G17 G90 G16;	Specifying the polar coordinate command and selecting the XY plane Setting the origin of the workpiece coordinate system as the origin of the polar coordinate system
N2 G81 X100.0 Y30.0 Z-20.0 R-5.0 F200.0 ;	
	Specifying a distance of 100 mm and an angle of 30 degrees
N3 G91 Y120.0 ;	Specifying a distance of 100 mm and an angle of +120 degrees
N4 Y120.0 ;	Specifying a distance of 100 mm and an angle of +120 degrees
N5 G15 G80 ;	Canceling the polar coordinate command

Limitation

- Specifying a radius in the polar coordinate mode

In the polar coordinate mode, specify a radius for circular interpolation or helical interpolation (G02, G03) with R.

- Axes that are not considered part of a polar coordinate command in the polar coordinate mode

Axes specified for the following commands are not considered part of the polar coordinate command:

- Dwell (G04)
- Programmable data input (G10)
- Local coordinate system setting (G52)
- Workpiece coordinate system setting (G92)
- Machine coordinate system setting (G53)
- Stored stroke check (G22)
- Coordinate system rotation (G68)
- Scaling (G51)

- Optional chamfering and corner R

Optional chamfering and corner R cannot be specified in polar coordinate mode.

5 FUNCTIONS TO SIMPLIFY PROGRAMMING

This chapter explains the following items:

5.1 CANNED CYCLE FOR DRILLING

Overview

Canned cycles for drilling make it easier for the programmer to create programs. With a canned cycle, a frequently-used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.
Table 5.1 (a) lists canned cycles for drilling.

Table 5.1 (a) Canned cycles for drilling

G code	Drilling (-Z direction)	Operation at the bottom of a hole	Retraction (+Z direction)	Application
G73	Intermittent feed	-	Rapid traverse	High-speed peck drilling cycle
G74	Feed	Dwell → Spindle CW	Feed	Left-hand tapping cycle
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle
G80	-	-	-	Cancel
G81	Feed	-	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	-	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell → Spindle CCW	Feed	Tapping cycle
G85	Feed	-	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back boring cycle
G88	Feed	Dwell → Spindle stop	Manual	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

Explanation

A canned cycle for drilling consists of a sequence of six operations.
Operation 1 ....Positioning of axes X and Y (including also another axis)
Operation 2 ....Rapid traverse up to point R level
Operation 3 ....Hole machining
Operation 4 ....Operation at the bottom of a hole
Operation 5 ....Retraction to point R level
Operation 6 ....Rapid traverse up to the initial point

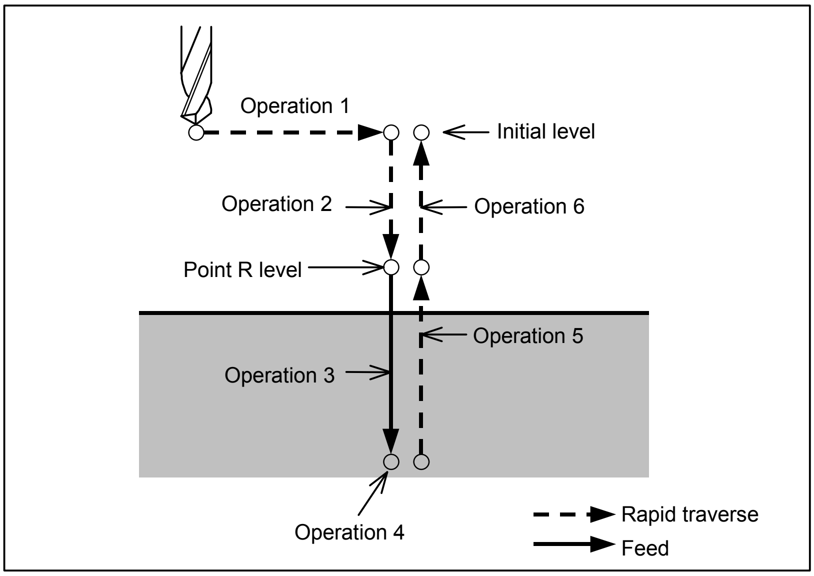


Fig. 5.1 (a) Operation sequence of canned cycle for drilling

- Positioning plane

The positioning plane is determined by plane selection code G17, G18, or G19.
The positioning axis is an axis other than the drilling axis.

- Drilling axis

Although canned cycles for drilling include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.
The drilling axis is a basic axis (X, Y, or Z) not used to define the positioning plane, or any axis parallel to that basic axis.
The axis (basic axis or parallel axis) used as the drilling axis is determined according to the axis address for the drilling axis specified in the same block as G codes G73 to G89.
If no axis address is specified for the drilling axis, the basic axis is assumed to be the drilling axis.

Table 5.1 (b) Positioning plane and drilling axis

G code	Positioning plane	Drilling axis
G17	Xp-Yp plane	Zp
G18	Zp-Xp plane	Yp
G19	Yp-Zp plane	Xp

Xp: X axis or an axis parallel to the X axis
Yp: Y axis or an axis parallel to the Y axis
Zp: Z axis or an axis parallel to the Z axis

Example

Assume that the U, V and W axes be parallel to the X, Y, and Z axes respectively. This condition is specified by parameter No. 1022.

G17 G81 Z _ _: The Z axis is used for drilling.
G17 G81 W _ _: The W axis is used for drilling.
G18 G81 Y _ _: The Y axis is used for drilling.
G18 G81 V _ _: The V axis is used for drilling.
G19 G81 X _ _: The X axis is used for drilling.
G19 G81 U _ _: The U axis is used for drilling.

G17 to G19 may be specified in a block in which any of G73 to G89 is not specified.

> ⚠ **CAUTION**
> Switch the drilling axis after canceling a canned cycle for drilling.

> **NOTE**
> A parameter FXY (No. 5101 #0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.

- Travel distance along the drilling axis G90/G91

The travel distance along the drilling axis varies for G90 and G91 as follows:

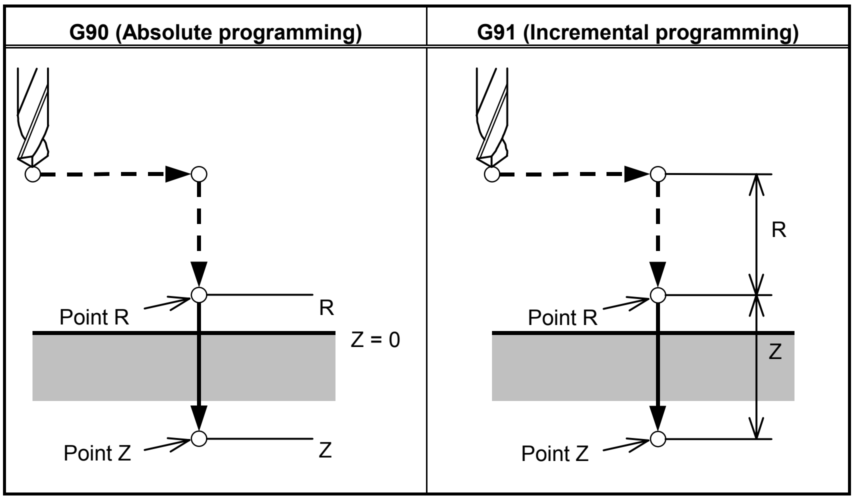


Fig. 5.1 (b) Absolute programming and incremental programming

- Drilling mode

G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode.

Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.

Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

- Return point level G98/G99

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.
The initial level does not change even when drilling is performed in the G99 mode.

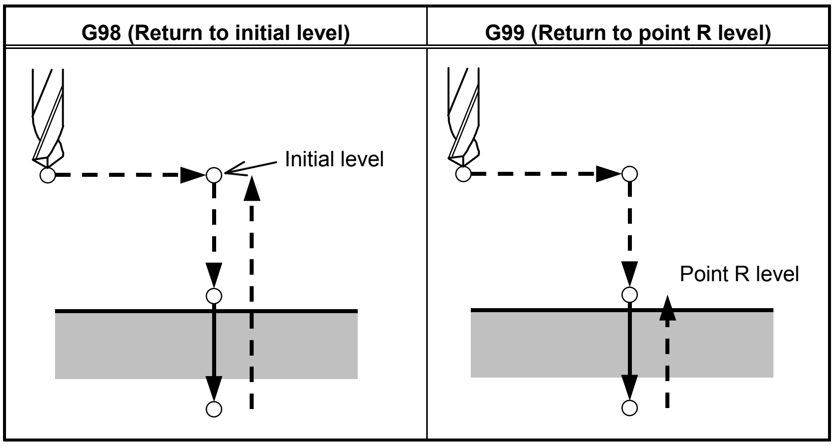


Fig. 5.1 (c) Initial level and point R level

- Repeat

To repeat drilling for equally-spaced holes, specify the number of repeats in K_.
K is effective only within the block where it is specified.
Specify the first hole position in incremental programming (G91).
If it is specified in absolute programming (G90), drilling is repeated at the same position.

Number of repeats K	The maximum command value = 9999

If K0 is specified, drilling data is stored, but drilling is not performed.

NOTE
For K, specify an integer of 0 or 1 to 9999.

- Cancel

To cancel a canned cycle, use G80 or a group 01 G code.

Group 01 G codes
G00 : Positioning (rapid traverse)
G01 : Linear interpolation
G02 : Circular interpolation or helical interpolation (CW)
G03 : Circular interpolation or helical interpolation (CCW)

- Symbols in figures

Subsequent sections explain the individual canned cycles. Figures in these Explanation use the following symbols:

	Positioning (rapid traverse G00)
	Cutting feed (linear interpolation G01)
	Manual feed
OSS	Oriented spindle stop (The spindle stops at a fixed rotation position)
	Shift (rapid traverse G00)
P	Dwell

5.1.1 High-Speed Peck Drilling Cycle (G73)

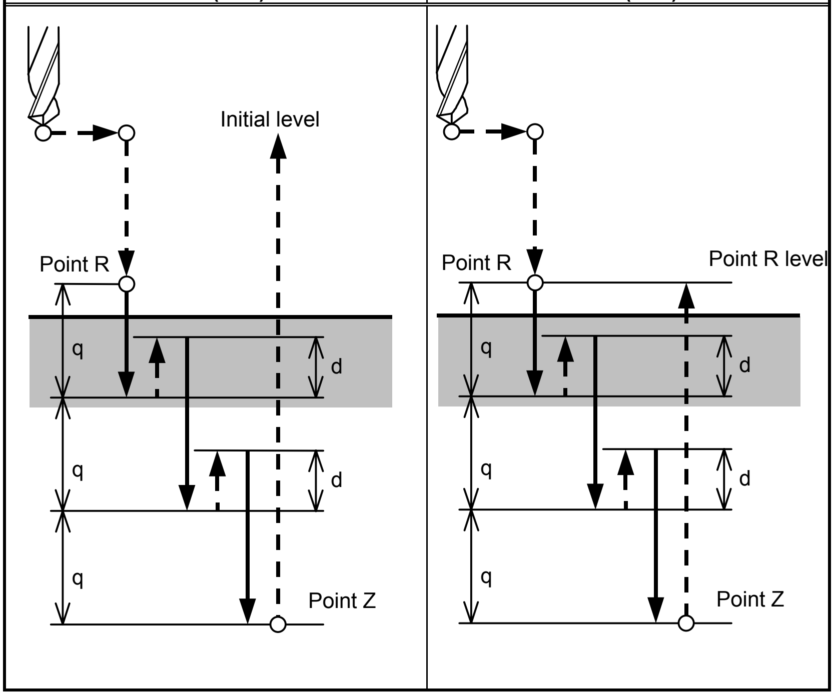
This cycle performs high-speed peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

Format

G73 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
Q_	: Depth of cut for each cutting feed
F_	: Cutting feedrate
K_	: Number of repeats (if required)

G73 (G98)	G73 (G99)



Explanation

- Operations

The high-speed peck drilling cycle performs intermittent feeding along the Z-axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows, drilling to be performed efficiently. Set the clearance, d, in parameter 5114.
The tool is retracted in rapid traverse.

- Spindle rotation

Before specifying G73, rotate the spindle using an auxiliary function (M code).

- Auxiliary function

When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- Q

Specify Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G73 in a single block. Otherwise, G73 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                  Cause the spindle to start rotating.
G90 G99 G73 X300. Y-250. Z-150. R-100. Q15. F120. ;
                            Position, drill hole 1, then return to point R.
Y-550. ;                    Position, drill hole 2, then return to point R.
Y-750. ;                    Position, drill hole 3, then return to point R.
X1000. ;                    Position, drill hole 4, then return to point R.
Y-550. ;                    Position, drill hole 5, then return to point R.
G98 Y-750. ;                Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;      Return to the reference position
M5 ;                        Cause the spindle to stop rotating.
```

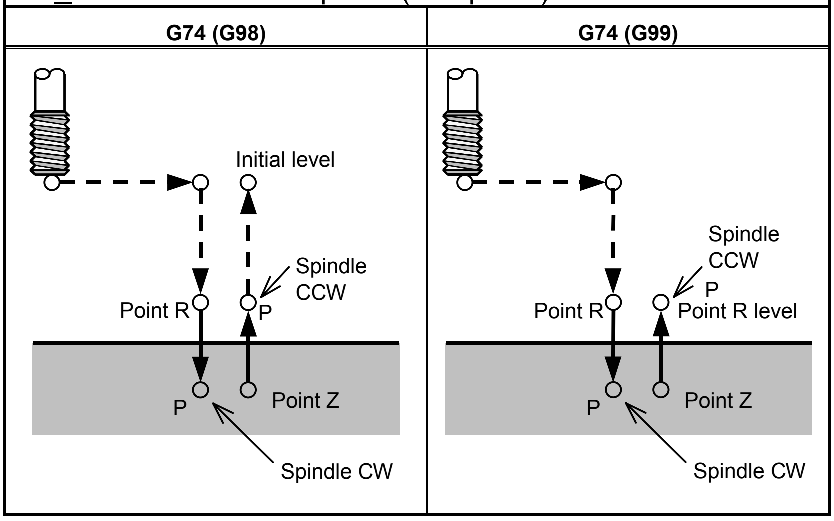
5.1.2 Left-Handed Tapping Cycle (G74)

This cycle performs left-handed tapping. In the left-handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Format

G74 X_ Y_ Z_ R_P_ F_ K_ ;

X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time
F_	: Cutting feedrate
K_	: Number of repeats (if required)



Explanation

- Operations

Tapping is performed by turning the spindle counterclockwise. When the bottom of the hole has been reached, the spindle is rotated clockwise for retraction. This creates a reverse thread.

> ⚠ **CAUTION**
> Feedrate overrides are ignored during left-handed tapping. A feed hold does not stop the machine until the return operation is completed.

- Spindle rotation

Before specifying G74, use an auxiliary function (M code) to rotate the spindle counterclockwise.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines,

the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G74 in a single block. Otherwise, G74 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M4 S100 ;	Cause the spindle to start rotating.
G90 G99 G74 X300. Y-250. Z-150. R-120. F120. ;	
	Position, tapping hole 1, then return to point R.
Y-550. ;	Position, tapping hole 2, then return to point R.
Y-750. ;	Position, tapping hole 3, then return to point R.
X1000. ;	Position, tapping hole 4, then return to point R.
Y-550. ;	Position, tapping hole 5, then return to point R.
G98 Y-750. ;	Position, tapping hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

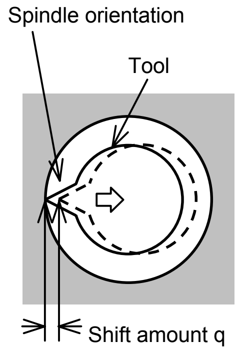
5.1.3 Fine Boring Cycle (G76)

The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

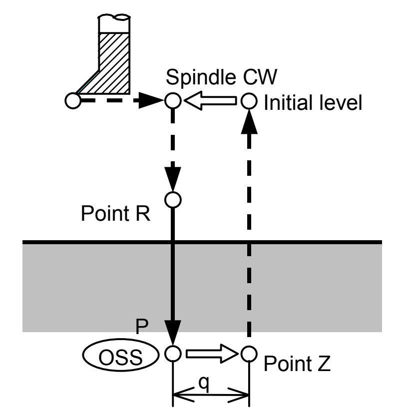
Format

G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

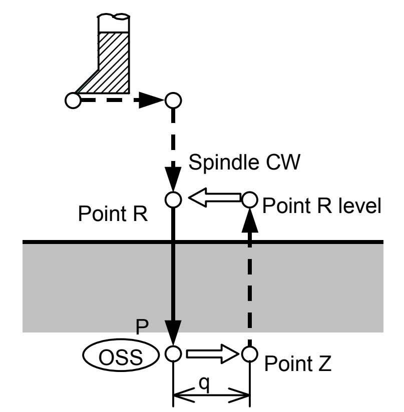
X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
Q_	: Shift amount at the bottom of a hole
P_	: Dwell time at the bottom of a hole
F_	: Cutting feedrate
K_	: Number of repeats (if required)



G76 (G98)



G76 (G99)



Explanation

- Operations

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool nose and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed.

- Spindle rotation

Before specifying G76, use a Auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any additional axes, drilling is not performed.

- P/Q

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in the parameter (No.5148).
Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.

> ⚠ **CAUTION**
> Q (shift at the bottom of a hole) is a modal value retained within canned cycles for drilling. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G76 in a single block. Otherwise, G76 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S500 ;	Cause the spindle to start rotating.
G90 G99 G76 X300. Y-250.	
	Position, bore hole 1, then return to point R.
Z-150. R-120. Q5.	Orient at the bottom of the hole, then shift by 5 mm.
P1000 F120. ;	Stop at the bottom of the hole for 1 s.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

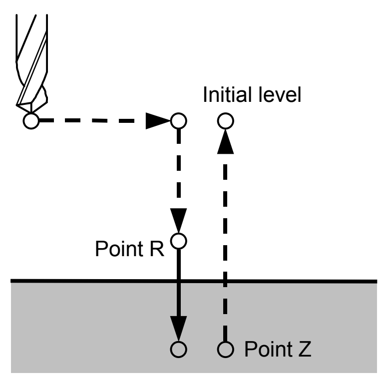
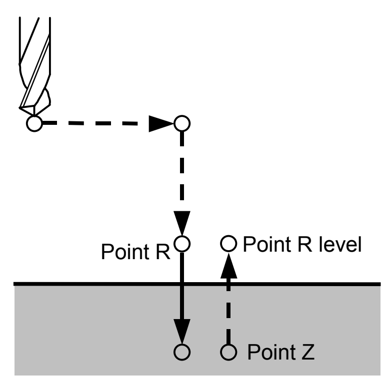
5.1.4 Drilling Cycle, Spot Drilling (G81)

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

G81 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
F_	:	Cutting feedrate
K_	:	Number of repeats (if required)

G81 (G98)	G81 (G99)
	

Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
The tool is then retracted in rapid traverse.

- Spindle rotation

Before specifying G81, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G81 in a single block. Otherwise, G81 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                  Cause the spindle to start rotating.
G90 G99 G81 X300. Y-250. Z-150. R-100. F120. ;
                            Position, drill hole 1, then return to point R.
Y-550. ;                    Position, drill hole 2, then return to point R.
Y-750. ;                    Position, drill hole 3, then return to point R.
X1000. ;                    Position, drill hole 4, then return to point R.
Y-550. ;                    Position, drill hole 5, then return to point R.
G98 Y-750. ;                Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;      Return to the reference position
M5 ;                        Cause the spindle to stop rotating.
```

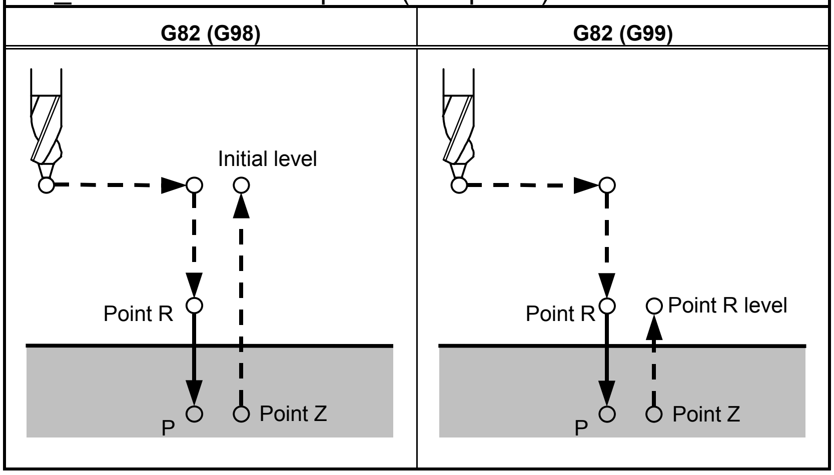
5.1.5 Drilling Cycle Counter Boring Cycle (G82)

This cycle is used for normal drilling.
Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse.
This cycle is used to drill holes more accurately with respect to depth.

Format

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time at the bottom of a hole
F_ : Cutting feed rate
K_ : Number of repeats (if required)



Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is then performed from point R to point Z.
When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.

- Spindle rotation

Before specifying G82, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G82 in a single block. Otherwise, G82 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                         Cause the spindle to start rotating.
G90 G99 G82 X300. Y-250. Z-150. R-100. P1000 F120. ;
                                   Position, drill hole 1, and dwell for 1 s at the
                                   bottom of the hole, then return to point R.
Y-550. ;                           Position, drill hole 2, then return to point R.
Y-750. ;                           Position, drill hole 3, then return to point R.
X1000. ;                           Position, drill hole 4, then return to point R.
Y-550. ;                           Position, drill hole 5, then return to point R.
G98 Y-750. ;                       Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0   ;           Return to the reference position
M5 ;                               Cause the spindle to stop rotating.
```

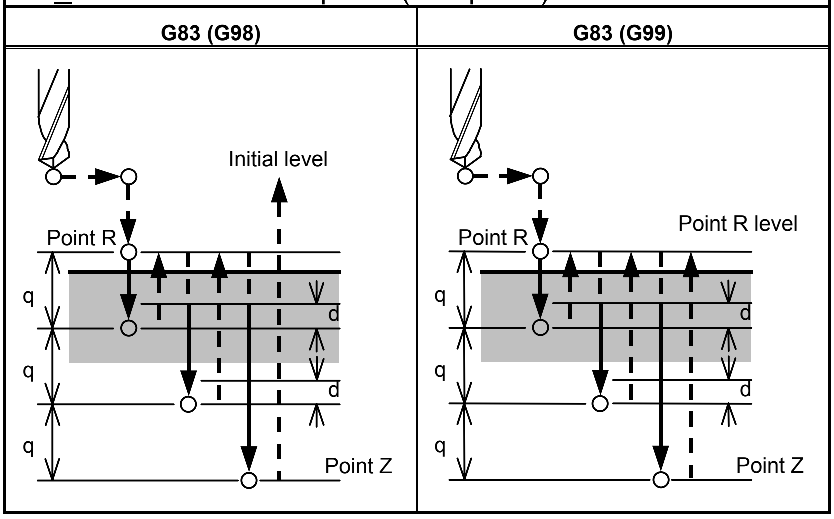
5.1.6 Peck Drilling Cycle (G83)

This cycle performs peck drilling.
It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Format

G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
Q_	: Depth of cut for each cutting feed
F_	: Cutting feedrate
K_	: Number of repeats (if required)



Explanation

- Operations

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.
In the second and subsequent cutting feeds, rapid traverse is performed up to a d point just before where the last drilling ended, and cutting feed is performed again. d is set in parameter (No.5115).
Be sure to specify a positive value in Q. Negative values are ignored.

- Spindle rotation

Before specifying G83, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- Q

Specify Q in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G83 in a single block. Otherwise, G83 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                 Cause the spindle to start rotating.
G90 G99 G83 X300. Y-250. Z-150. R-100. Q15. F120. ;
                           Position, drill hole 1, then return to point R.
Y-550. ;                   Position, drill hole 2, then return to point R.
Y-750. ;                   Position, drill hole 3, then return to point R.
X1000. ;                   Position, drill hole 4, then return to point R.
Y-550. ;                   Position, drill hole 5, then return to point R.
G98 Y-750. ;               Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;     Return to the reference position
M5 ;                       Cause the spindle to stop rotating.
```

5.1.7 Small-Hole Peck Drilling Cycle

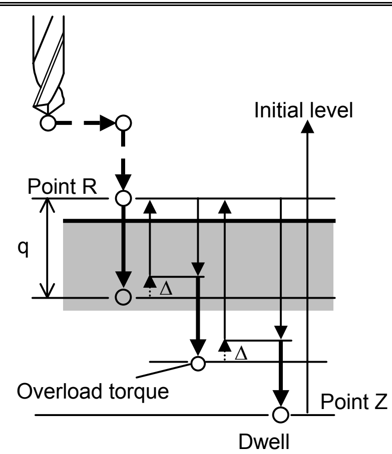
An arbor with the overload torque detection function is used to retract the tool when the overload torque detection signal (skip signal) is detected during drilling. Drilling is resumed after the spindle speed and cutting feedrate are changed. These steps are repeated in this peck drilling cycle.
The mode for the small–hole peck drilling cycle is selected when the M code in parameter 5163 is specified. The cycle can be started by specifying G83 in this mode. This mode is canceled when G80 is specified or when a reset occurs.

Format

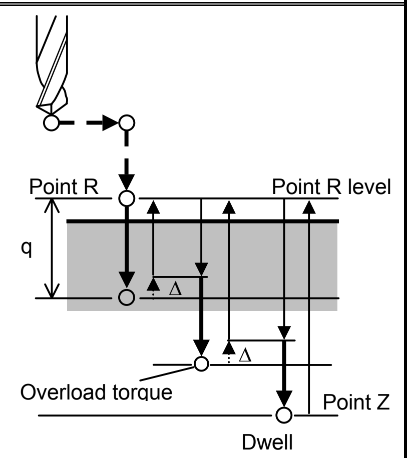
G83 X_ Y_ Z_ R_ Q_ F_I_ K_P ;

X_ Y_ : Hole position data
Z_ : Distance from point R to the bottom of the hole
R_ : Distance from the initial level to point R
Q_ : Depth of each cut
F_ : Cutting feedrate
I_ : Forward or backward traveling speed (same format as F above)
(If this is omitted, the values in parameters No.5172 and No.5173 are assumed as defaults.)
K_ : Number of times the operation is repeated (if required)
P_ : Dwell time at the bottom of the hole
(If this is omitted, P0 is assumed as the default.)

G83 (G98)



G83 (G99)



Δ: Initial clearance when the tool is retracted to point R and the clearance from the bottom of the hole in the second or subsequent drilling (parameter 5174)
q: Depth of each cut
Path along which the tool travels at the rapid traverse rate
Path along which the tool travels at the programmed cutting feedrate
Path along which the tool travels at the forward or backward rate during the cycle specified with parameters

Explanations

- Componet operations of the cycle

* X- and Y-axis positioning
* Positioning at point R along the Z–axis
* Cutting along the Z-axis (first time, depth of cut Q, incremental)

Repeated until point Z is reached

Retracting
(bottom of hole → minimum clearance Δ, incremental)
Retraction
(bottom of hole+Δ→ to point R, absolute)
Forwarding
(point R → to point with hole bottom + clearance Δ, absolute)
Cutting
(second and subsequent times, cut of depth Q + Δ, incremental)

* Dwell
* Return to point R along the Z-axis (or initial point) = end of cycle

Acceleration/deceleration during advancing and retraction is controlled according to the cutting feed acceleration/deceleration time constant.
When retraction is performed, the position is checked at point R.

- Specifying an M code

When the M code in parameter 5163 is specified, the system enters the mode for the small–hole peck drilling cycle.
This M code does not wait for FIN. Care must be taken when this M code is specified with another M code in the same block.
(Example) M03 M□□ ; → Waits for FIN.
M□□ M03 ; → Does not wait for FIN.

- Specifying a G code

When G83 is specified in the mode for the small-hole peck drilling cycle, the cycle is started.
This continuous–state G code remains unchanged until another canned cycle is specified or until the G code for canceling the canned cycle is specified. This eliminates the need for specifying drilling data in each block when identical drilling is repeated.

- Signal indicating that the cycle is in progress

In this cycle, the signal indicating that the small–hole peck drilling cycleis in progress is output after the tool is positioned at the hole position along the axes not used for drilling. Signal output continues during positioning to point R along the drilling axis and terminates upon a return to point R or the initial level. For details, refer to the manual of the machine tool builder.

- Overload torque detection signal

A skip signal is used as the overload torque detection signal. The skip signal is effective while the tool is advancing or drilling and the tool

tip is between points R and Z. (The signal causes a retraction). For details, refer to the manual of the machine tool builder.

> **NOTE**
> When receiving overload torque detect signal while the tool is advancing, the tool will be retracted (clearance Δ and to the point R), then advanced to the same target point as previous advancing.

- Changing the drilling conditions

In a single G83 cycle, drilling conditions are changed for each drilling operation (advance → drilling → retraction). Bits 1 and 2 of parameter OLS, NOL No. 5160 can be specified to suppress the change in drilling conditions.

1 Changing the cutting feedrate
The cutting feedrate programmed with the F code is changed for each of the second and subsequent drilling operations. In parameters No.5166 and No.5167, specify the respective rates of change applied when the skip signal is detected and when it is not detected in the previous drilling operation.

Cutting feedrate = F × α

<First drilling> α=1.0
<Second or subseqent drilling> $\alpha=\alpha\times\beta\div100$, where β is the rate of change for each drilling operation
When the skip signal is detected during the previous drillingoperation: β=b1% (parameter No.5166)
When the skip signal is not detected during the previous drillingoperation: β=b2% (parameter No.5167)

If the rate of change in cutting feedrate becomes smaller than the rate specified in parameter 5168, the cutting feedrate is not changed.
The cutting feedrate can be increased up to the maximum cutting feedrate.

2 Changing the spindle speed
The spindle speed programmed with the S code is changed for each of the second and subsequent advances. In parameters 5164 and 5165, specify the rates of change applied when the skip signal is detected and when it is not detected in the previous drilling operation.

Spindle speed = S × γ

<First drilling> γ=1.0
<Second or subseqent drilling> $\gamma=\gamma\times\delta\div100$, where δ is the rate of change for each drilling operation

When the skip signal is detected during the previous drilling operation: δ=d1% (parameter No.5164)
When the skip signal is not detected during the previous drilling operation: δ=d2% (parameter No.5165)

When the cutting feedrate reaches the minimum rate, the spindle speed is not changed. The spindle speed can be increased up to a value corresponding to the maximum value of S analog data.

- Advance and retraction

Advancing and retraction of the tool are not executed in the same manner as rapid-traverse positioning. Like cutting feed, the two operations are carried out as interpolated operations. Note that the tool life management function excludes advancing and retraction from the calculation of the tool life.

- Specifying addess I

The forward or backward traveling speed can be specified with address I in the same format as address F, as shown below:

G83 I1000 ; (without decimal point)
G83 I1000. ; (with decimal point)

Both commands indicate a speed of 1000 mm/min.

Address I specified with G83 in the continuous-state mode continues to be valid until G80 is specified or until a reset occurs.

NOTE
If address I is not specified and parqmeter No.5172 (for backword) or No.5173 (for forword) is set to 0, the forword or backword travel speed is same as the cutting feedrate specified by F.

- Fuctions that can be specified

In this canned cycle mode, the following functions can be specified:

- Hole position on the X-axis, Y-axis, and additional axis
- Operation and branch by custom macro
- Subprogram (hole position group, etc.) calling
- Switching between absolute and incremental modes
- Coordinate system rotation
- Scaling (This command will not affect depth of cut Q or small clearance Δ.)
- Dry run
- Feed hold

- Single block

When single-block operation is enabled, drilling is stopped after each retraction. Also, a single block stop is performed by setting parameter SBC (No.5105 bit 0)

- Feedrate override

The feedrate override function works during cutting, retraction, and advancing in the cycle.

- Custom macro interface

The number of retractions made during cutting and the number of retractions made in response to the overload signal received during cutting can be output to custom macro common variables (#100 to #149) specified in parameters No.5170 and No.5171. Parameters No.5170 and No.5171 can specify variable numbers within the range of #100 to #149.

Parameter No.5170: Specifies the number of the common variable to which the number of retractions made during cutting is output.

Parameter No.5171: Specifies the number of the common variable to which the number of retractions made in response to the overload signal received during cutting is output.

> **NOTE**
> The numbers of retruction output to common valiables are cleared by G83 while small-hole peck driling cycle mode.

Limitation

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P_ in an independent block.

Example

M03 S_ ; Cause the spindle to start rotating.
M□□ ; Specifies the small-hole peck drilling cycle mode.
G83 X_ Y_ Z_ R_ Q_ F_ I_ K_ P_ ;
Specifies the small-hole peck drilling cycle.
X_ Y_ ; Drills at another position.
:
:
G80 ; Cancels the small-hole peck drilling cycle mode.

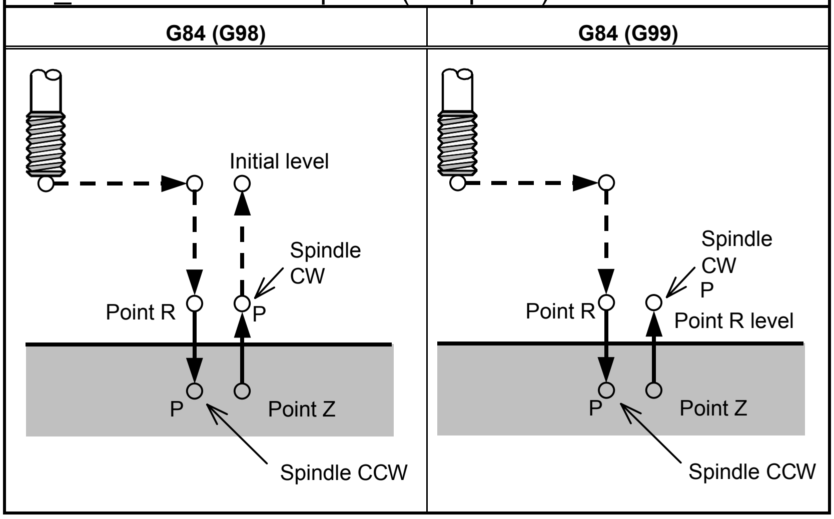
5.1.8 Tapping Cycle (G84)

This cycle performs tapping.
In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format

G84 X_ Y_ Z_ R_P_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
P_ : Dwell time
F_ : Cutting feedrate
K_ : Number of repents (if required)



Explanation

- Operations

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.

> ⚠ **CAUTION**
> Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.

- Spindle rotation

Before specifying G84, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation,

without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When the K is used to specify number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G84 in a single block. Otherwise, G84 will be canceled.

Example

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G84 X300. Y-250. Z-150. R-120. P300 F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

5.1.9 Boring Cycle (G85)

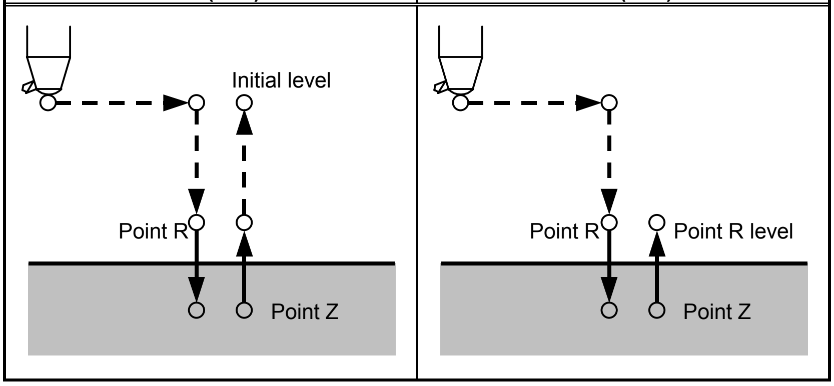
This cycle is used to bore a hole.

Format

G85 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
F_ : Cutting feed rate
K_ : Number of repeats (if required)

G85 (G98)	G85 (G99)



Explanation

- Operations

After positioning along the X- and Y- axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
When point Z has been reached, cutting feed is performed to return to point R.

- Spindle rotation

Before specifying G85, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G85 in a single block. Otherwise, G85 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S100 ;                 Cause the spindle to start rotating.
G90 G99 G85 X300. Y-250. Z-150. R-120. F120. ;
                          Position, drill hole 1, then return to point R.
Y-550. ;                  Position, drill hole 2, then return to point R.
Y-750. ;                  Position, drill hole 3, then return to point R.
X1000. ;                  Position, drill hole 4, then return to point R.
Y-550. ;                  Position, drill hole 5, then return to point R.
G98 Y-750. ;              Position, drill hole 6, then return to the
                          initial level.
G80 G28 G91 X0 Y0 Z0   ;  Return to the reference position
M5 ;                      Cause the spindle to stop rotating.
```

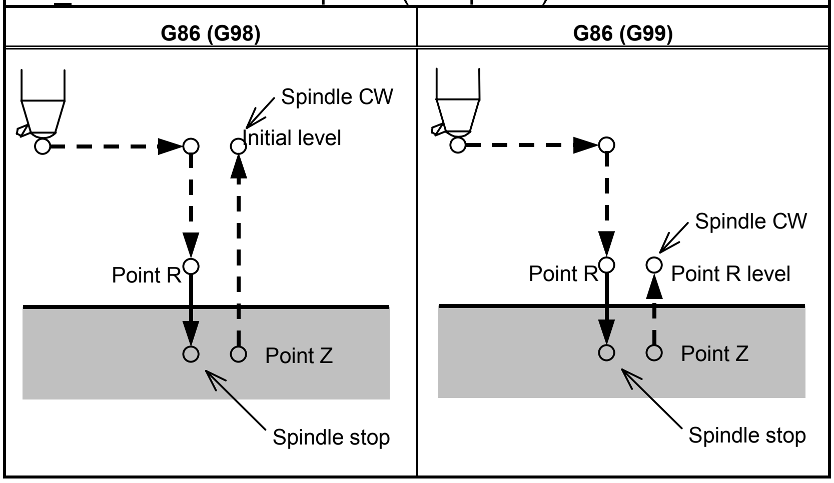
5.1.10 Boring Cycle (G86)

This cycle is used to bore a hole.

Format

G86 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)



Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.

- Spindle rotation

Before specifying G86, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G86 in a single block. Otherwise, G86 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                      Cause the spindle to start rotating.
G90 G99 G86 X300. Y-250. Z-150. R-100. F120. ;
                                Position, drill hole 1, then return to point R.
Y-550. ;                        Position, drill hole 2, then return to point R.
Y-750. ;                        Position, drill hole 3, then return to point R.
X1000. ;                        Position, drill hole 4, then return to point R.
Y-550. ;                        Position, drill hole 5, then return to point R.
G98 Y-750. ;                    Position, drill hole 6, then return to the initial
                                level.
G80 G28 G91 X0 Y0 Z0   ;        Return to the reference position
M5 ;                            Cause the spindle to stop rotating.
```

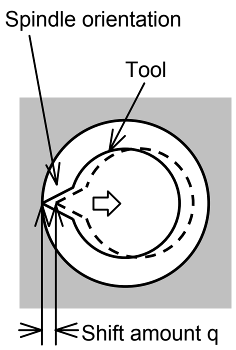
5.1.11 Back Boring Cycle (G87)

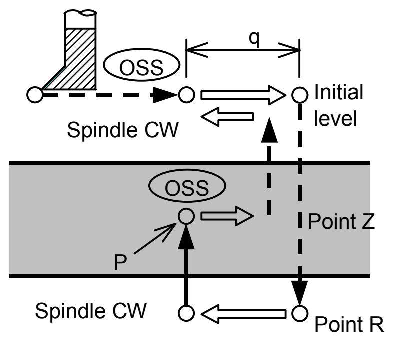
This cycle performs accurate boring.

Format

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R
Q_ : Shift amount at the bottom of a hole
P_ : Dwell time at the bottom of a hole
F_ : Cutting feed rate
K_ : Number of repeats (if required)



G87 (G98)	G87 (G99)
	Not used

Explanation

After positioning along the X- and Y-axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool nose, positioning (rapid traverse) is performed to the bottom of the hole (point R).
The tool is then shifted in the direction of the tool nose and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z-axis until point Z is reached.
At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool nose, then the tool is returned to the initial level. The tool is then shifted in the direction of the tool nose and the spindle is rotated clockwise to proceed to the next block operation.

- Spindle rotation

Before specifying G87, use an auxiliary function (M code) to rotate the spindle.
If drilling is continuously performed with a small value specified for the distance between the hole position and point R level or between the initial level and point R level, the normal spindle speed may not be reached at the start of hole cutting operation. In this case, insert a dwell before each drilling operation with G04 to delay the operation, without specifying the number of repeats for K. For some machines, the above note may not be considered. Refer to the manual provided by the machine tool builder.

- Auxiliary function

When the G87 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any additional axes, drilling is not performed.

- P/Q

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in the parameter (No. 5148).
Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.

> ⚠ **CAUTION**
> Q (shift at the bottom of a hole) is a modal value retained in canned cycles for drilling. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G87 in a single block. Otherwise, G87 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

M3 S500 ;	Cause the spindle to start rotating.
G90 G87 X300. Y-250.	Position, bore hole 1.
Z-150. R-120. Q5.	Orient at the initial level, then shift by 5 mm.
P1000 F120. ;	Stop at point Z for 1 s.
Y-550. ;	Position, drill hole 2.
Y-750. ;	Position, drill hole 3.
X1000. ;	Position, drill hole 4.
Y-550. ;	Position, drill hole 5.
Y-750. ;	Position, drill hole 6
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position
M5 ;	Cause the spindle to stop rotating.

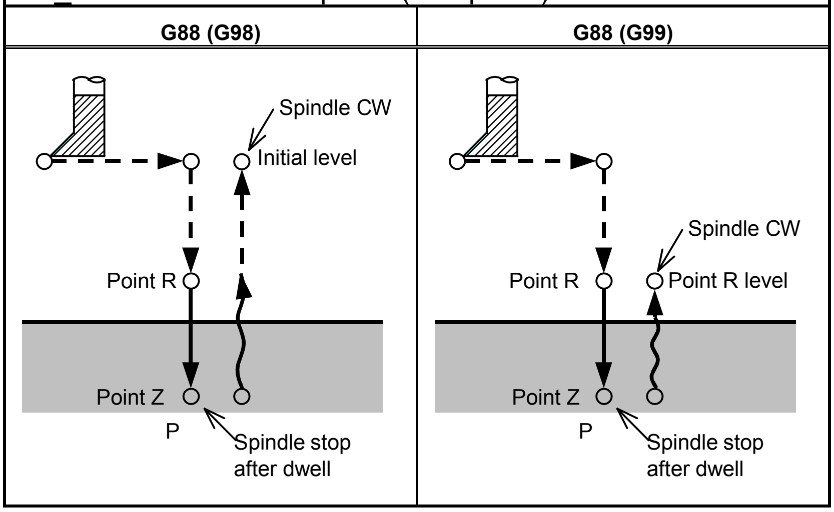
5.1.12 Boring Cycle (G88)

This cycle is used to bore a hole.

Format

G88 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time at the bottom of a hole
F_	: Cutting feed rate
K_	: Number of repeats (if required)



Explanation

- Operations

After positioning along the X- and Y-axes, rapid traverse is performed to point R. Boring is performed from point R to point Z.
When boring is completed, a dwell is performed at the bottom of the hole, then the spindle is stopped and enters the hold state. At this time, you can switch to the manual mode and move the tool manually. Any manual operations are available; it is desirable to finally retract the tool from the hole for safety, though.
At the restart of machining in the DNC operation or memory mode, the tool returns to the initial level or point R level according to G98 or G99 and the spindle rotates clockwise. Then, operation is restarted according to the programmed commands in the next block.

- Spindle rotation

Before specifying G88, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G88 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G88 in a single block. Otherwise, G88 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S2000 ;                    Cause the spindle to start rotating.
G90 G99 G88 X300. Y-250. Z-150. R-100. P1000 F120. ;
                              Position, drill hole 1, return to point R then stop
                              at the bottom of the hole for 1 s.
Y-550. ;                      Position, drill hole 2, then return to point R.
Y-750. ;                      Position, drill hole 3, then return to point R.
X1000. ;                      Position, drill hole 4, then return to point R.
Y-550. ;                      Position, drill hole 5, then return to point R.
G98 Y-750. ;                  Position, drill hole 6, then return to the initial
                              level.
G80 G28 G91 X0 Y0 Z0   ;      Return to the reference position
M5 ;                          Cause the spindle to stop rotating.
```

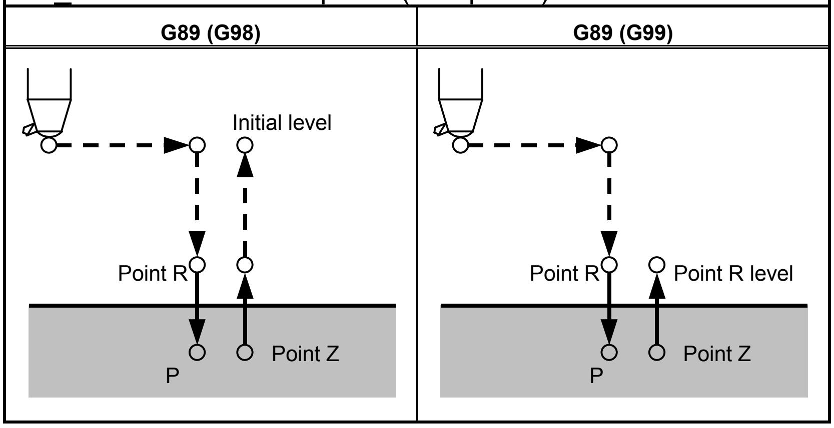
5.1.13 Boring Cycle (G89)

This cycle is used to bore a hole.

Format

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats (if required)



Explanation

- Operations

This cycle is almost the same as G85. The difference is that this cycle performs a dwell at the bottom of the hole.

- Spindle rotation

Before specifying G89, use an auxiliary function (M code) to rotate the spindle.

- Auxiliary function

When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

- Tool length compensation

When a tool length compensation (G43, G44, or G49) is specified in the canned cycle for drilling, the offset is applied after the time of positioning to point R.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle for drilling must be canceled.

- Drilling

In a block that does not contain X, Y, Z, R, or any other axes, drilling is not performed.

- P

Specify P in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03) and G89 in a single block. Otherwise, G89 will be canceled.

- Tool offset

In the canned cycle mode for drilling, tool offsets are ignored.

Example

```
M3 S100 ;                 Cause the spindle to start rotating.
G90 G99 G89 X300. Y-250. Z-150. R-120. P1000 F120. ;
                          Position, drill hole 1, return to point R then stop
                          at the bottom of the hole for 1 s.
Y-550. ;                  Position, drill hole 2, then return to point R.
Y-750. ;                  Position, drill hole 3, then return to point R.
X1000. ;                  Position, drill hole 4, then return to point R.
Y-550. ;                  Position, drill hole 5, then return to point R.
G98 Y-750. ;              Position, drill hole 6, then return to the initial
                          level.
G80 G28 G91 X0 Y0 Z0  ;   Return to the reference position
M5 ;                      Cause the spindle to stop rotating.
```

5.1.14 Canned Cycle Cancel for Drilling (G80)

G80 cancels canned cycles for drilling.

Format

G80 ;

Explanation

All canned cycles for drilling are canceled to perform normal operation. Point R and point Z are cleared.
Other drilling data is also canceled (cleared).

Example

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G88 X300. Y-250. Z-150. R-120. F120. ;	
	Position, drill hole 1, then return to point R.
Y-550. ;	Position, drill hole 2, then return to point R.
Y-750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y-550. ;	Position, drill hole 5, then return to point R.
G98 Y-750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position, canned cycle cancel
M5 ;	Cause the spindle to stop rotating.

5.1.15 Example for Using Canned Cycles for Drilling

Offset value +200.0 is set in offset No.11, +190.0 is set in offset No.15, and +150.0 is set in offset No.31

Program example

;		
N001	G92 X0 Y0 Z0;	Coordinate setting at reference position
N002	G90 G00 Z250.0 T11 M6;	Tool change
N003	G43 Z0 H11;	Initial level, tool length compensation
N004	S30 M3;	Spindle start
N005	G99 G81 X400.0 Y-350.0 Z-153.0 R-97.0 F120;	Positioning, then #1 drilling
N006	Y-550.0;	Positioning, then #2 drilling and point R level return
N007	G98 Y-750.0;	Positioning, then #3 drilling and initial level return
N008	G99 X1200.0;	Positioning, then #4 drilling and point R level return
N009	Y-550.0;	Positioning, then #5 drilling and point R level return
N010	G98 Y-350.0;	Positioning, then #6 drilling and initial level return
N011	G00 X0 Y0 M5;	Reference position return, spindle stop
N012	G49 Z250.0 T15 M6;	Tool length compensation cancel, tool change
N013	G43 Z0 H15;	Initial level, tool length compensation
N014	S20 M3;	Spindle start
N015	G99 G82 X550.0 Y-450.0 Z-130.0 R-97.0 P300 F70;	Positioning, then #7 drilling, point R level return
N016	G98 Y-650.0;	Positioning, then #8 drilling, initial level return
N017	G99 X1050.0;	Positioning, then #9 drilling, point R level return
N018	G98 Y-450.0;	Positioning, then #10 drilling, initial level return
N019	G00 X0 Y0 M5;	Reference position return, spindle stop
N020	G49 Z250.0 T31 M6;	Tool length compensation cancel, tool change
N021	G43 Z0 H31;	Initial level, tool length compensation
N022	S10 M3;	Spindle start
N023	G85 G99 X800.0 Y-350.0 Z-153.0 R47.0 F50;	Positioning, then #11 drilling, point R level return
N024	G91 Y-200.0 K2;	Positioning, then #12, 13 drilling, point R level return
N025	G28 X0 Y0 M5;	Reference position return, spindle stop
N026	G49 Z0;	Tool length compensation cancel
N027	M0;	Program stop

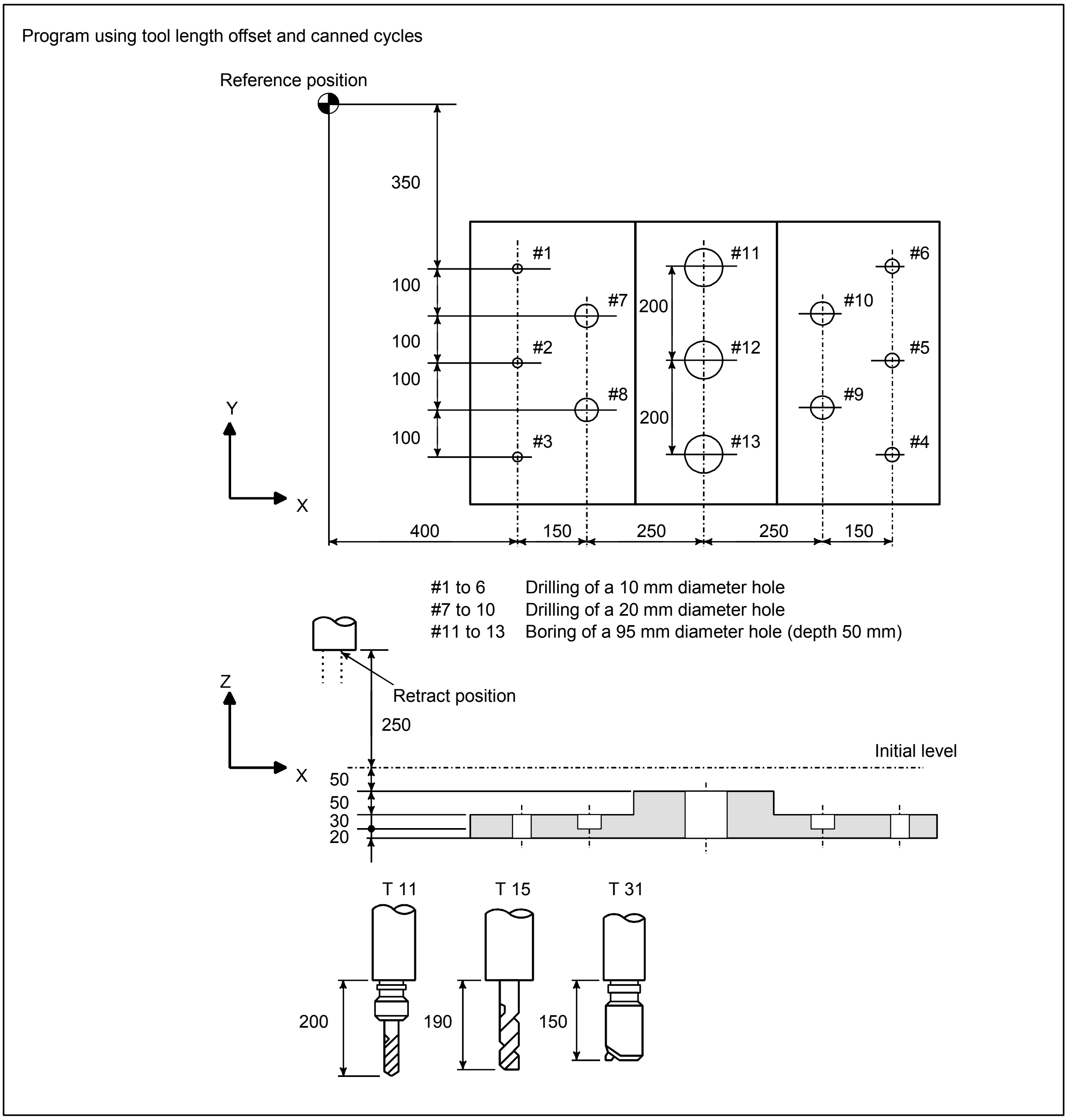


Fig. 5.1.15 (a) Example for using canned cycles for drilling

5.2 RIGID TAPPING

The tapping cycle (G84) and left-handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode.
In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using auxiliary functions M03 (rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping.
In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle.
When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.
Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

5.2.1 Rigid Tapping (G84)

When the spindle motor is controlled in rigid mode as if it were a servo motor, a tapping cycle can be sped up.

Format

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

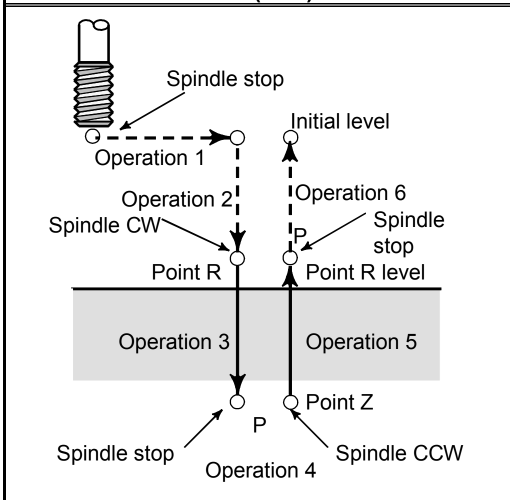
X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of the hole and at point R when a return is made
F_	:	Cutting feedrate
K_	:	Number of repeats (Only for necessity of repeat)

G84.2 X_ Y_ Z_ R_ P_ F_ L_ ;

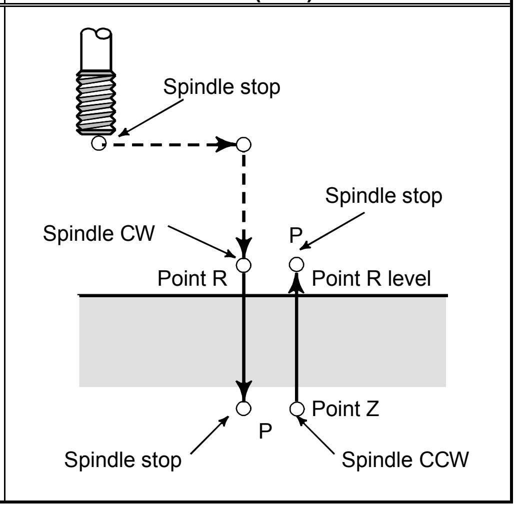
(Series 15 format)

L_	:	Number of repeats (only for necessity of repeat)

G84 (G98)



G84 (G99)



Explanation

After positioning along the X- and Y-axes, rapid traverse is performed to point R.

Tapping is performed from point R to point Z. When tapping is completed, the spindle is stopped and a dwell is performed. The spindle is then rotated in the reverse direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed.

While tapping is being performed, the feedrate override and spindle override are assumed to be 100%. Feedrate override can be enabled by setting, however.

- Rigid mode

Rigid mode can be specified using any of the following methods:
- Specify M29 S***** before a tapping command.
- Specify M29 S***** in a block which contains a tapping command.
- Specify G84 for rigid tapping (parameter G84 No. 5200 #0 set to 1).

- Thread lead

In feed-per-minute mode, the thread lead is obtained from the expression, feedrate ÷ spindle speed. In feed-per-revolution mode, the thread lead equals the feedrate speed.

- Tool length compensation

If a tool length compensation (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

- Series 15 format command

Rigid tapping can be performed using Series 15 format commands. The rigid tapping sequence (including data transfer to and from the PMC), Limitation, and the like are the same as described in this chapter.

- Acceleration/deceleration after interpolation

Linear or bell-shaped acceleration/deceleration can be applied.

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

- Override

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:
- Extraction override
- Override signal

Details are given later.

- Dry run

Dry run can be executed also in G84 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G74).
When G84 (G74) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

- Reset

When a reset is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G74) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

When bit 6 (FHD) of parameter No. 5200 is set to 0, feed hold and single block are invalid in the G84 (G74) mode. When this bit is set to 1, they are valid.

- Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."
With other manual operations, rigid tapping cannot be performed.

- Backlash compensation

In the rigid tapping mode, backlash compensation is applied to compensate the lost motion when the spindle rotates clockwise or counterclockwise. Set the amount of backlash in parameters Nos. 5321 to 5324.
Along the drilling axis, backlash compensation has been applied.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm PS0206 is issued.

- S command

- If a speed higher than the maximum speed for the gear being used is specified, alarm PS0200 is issued.
- When the rigid tapping canned cycle is cancelled, the S command used for rigid tapping is cleared to S0.

- Distribution amount for the spindle

The maximum distribution amount is as follows (displayed on diagnostic screen No. 451):

- For a serial spindle: 32,767 pulses per 8 ms

This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

- F command

If a value exceeding the upper limit of cutting feedrate is specified, alarm PS0011 is issued.

- Unit of F command

	Metric input	Inch input	Remarks
G94	1 mm/min	0.01 inch/min	Decimal point programming allowed
G95	0.01 mm/rev	0.0001 inch/rev	Decimal point programming allowed

- M29

If an S command and axis movement are specified between M29 and G84, alarm PS0203 is issued. If M29 is specified in a tapping cycle, alarm PS0204 is issued.

- P

Specify P in a block that performs drilling. If P is specified in a non-drilling block, it is not stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03 or G60 (when the MDL bit (bit 0 of parameter 5431) is set to 1)) and G74 in a single block. Otherwise, G74 will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Program restart

A program cannot be restarted during rigid tapping.

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P_ in an independent block.

Example

Z-axis feedrate 1000 mm/min
Spindle speed 1000 min^{-1}
Thread lead 1.0 mm

<Programming of feed per minute>

```
G94;                          Specify a feed-per-minute command.
G00 X120.0 Y100.0 ;           Positioning
M29 S1000 ;                   Rigid mode specification
G84 Z-100.0 R-20.0 F1000 ;    Rigid tapping
```

<Programming of feed per revolution>

```
G95 ;                         Specify a feed-per-revolution command.
G00 X120.0 Y100.0 ;           Positioning
M29 S1000 ;                   Rigid mode specification
G84 Z-100.0 R-20.0 F1.0 ;     Rigid tapping
```

5.2.2 Left-Handed Rigid Tapping Cycle (G74)

When the spindle motor is controlled in rigid mode as if it were a servo motor, tapping cycles can be speed up.

Format

G74 X_ Y_ Z_ R_ P_ F_ K_ ;

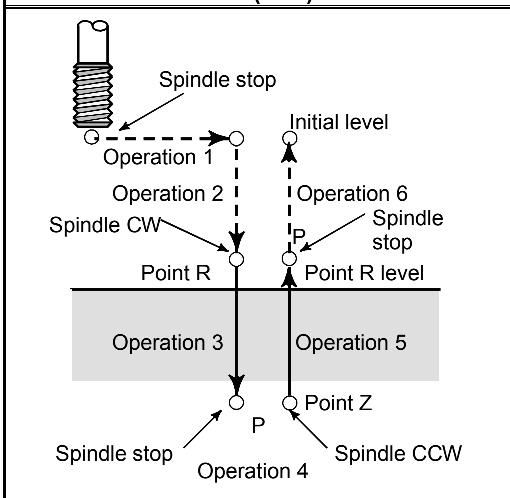
- X_ Y_ : Hole position data
- Z_ : The distance from point R to the bottom of the hole and the position of the bottom of the hole
- R_ : The distance from the initial level to point R level
- P_ : Dwell time at the bottom of the hole and at point R when return is made.
- F_ : Cutting feedrate
- K_ : Number of repeats (Only for necessity of repeat)

G84.2 X_ Y_ Z_ R_ P_ F_ L_ ;

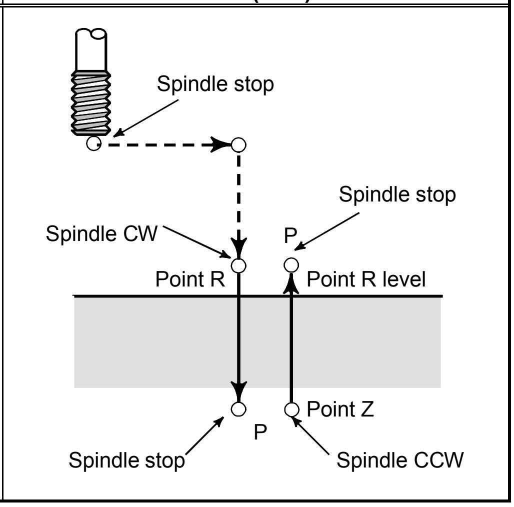
(Series 15 format)

- L_ : Number of repeats (Only for necessity of repeat)

G74 (G98)



G74 (G99)



Explanation

After positioning along the X- and Y-axes, rapid traverse is performed to point R.

Tapping is performed from point R to point Z. When tapping is completed, the spindle is stopped and a dwell is performed. The spindle is then rotated in the normal direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed.

While tapping is being performed, the feedrate override and spindle override are assumed to be 100%. Feedrate override can be enabled by setting, however.

- Rigid mode

Rigid mode can be specified using any of the following methods:

- Specify M29 S***** before a tapping command.
- Specify M29 S***** in a block which contains a tapping command.
- Specify G74 for rigid tapping. (parameter G84 (No. 5200#0) set to1).

- Thread lead

In feed-per-minute mode, the thread lead is obtained from the expression, feedrate ÷ spindle speed. In feed-per-revolution mode, the thread lead equals the feedrate.

- Tool length compensation

If a tool length compensation (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

- Series 15 format command

Rigid tapping can be performed using Series 15 format commands. The rigid tapping sequence (including data transfer to and from the PMC), Limitation, and the like are the same as described in this chapter.

- Acceleration/deceleration after interpolation

Linear or bell-shaped acceleration/deceleration can be applied.

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

- Override

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:

- Extraction override
- Override signal

Details are given later.

- Dry run

Dry run can be executed also in G84 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G74).
When G84 (G74) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

- Reset

When a reset is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G74) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

When bit 6 (FHD) of parameter No. 5200 is set to 0, feed hold and single block are invalid in the G84 (G74) mode. When this bit is set to 1, they are valid.

- Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."
With other manual operations, rigid tapping cannot be performed.

- Backlash compensation

In the rigid tapping mode, backlash compensation is applied to compensate the lost motion when the spindle rotates clockwise or counterclockwise. Set the amount of backlash in parameters Nos. 5321 to 5324.
Along the drilling axis, backlash compensation has been applied.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm PS0206 is issued.

- S command

- Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm PS0200.
- When the rigid tapping canned cycle is cancelled, the S command used for rigid tapping is cleared to S0.

- Distribution amount for the spindle

The maximum distribution amount is as follows (displayed on diagnostic screen No. 451):

- For a serial spindle: 32,767 pulses per 8 ms

This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

- F command

Specifying a value that exceeds the upper limit of cutting feedrate causes alarm PS0011.

- Unit of F command

	Metric input	Inch input	Remarks
G94	1 mm/min	0.01 inch/min	Decimal point programming allowed
G95	0.01 mm/rev	0.0001 inch/rev	Decimal point programming allowed

- M29

Specifying an S command or axis movement between M29 and G84 causes alarm PS0203.
Then, specifying M29 in the tapping cycle causes alarm PS0204.

- P

Specify P in a block that performs drilling. If P is specified in a non-drilling block, it is not stored as modal data.

- Cancel

Do not specify a G code of the 01 group (G00 to G03 or G60 (when the MDL bit (bit 0 of parameter 5431) is set to 1)) and G74 in a single block. Otherwise, G74 will be canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P_ in an independent block.

Example

Z-axis feedrate 1000 mm/min
Spindle speed 1000 min^{-1}
Thread lead 1.0 mm

<Programming for feed per minute>

```
G94 ;                          Specify a feed-per-minute command.
G00 X120.0 Y100.0 ;            Positioning
M29 S1000 ;                    Rigid mode specification
G74 Z-100.0 R-20.0 F1000 ;     Rigid tapping
```

<Programming for feed per revolution>

```
G95 ;                          Specify a feed-per-revolution command.
G00 X120.0 Y100.0 ;            Positioning
M29 S1000 ;                    Rigid mode specification
G74 Z-100.0 R-20.0 F1.0 ;      Rigid tapping
```

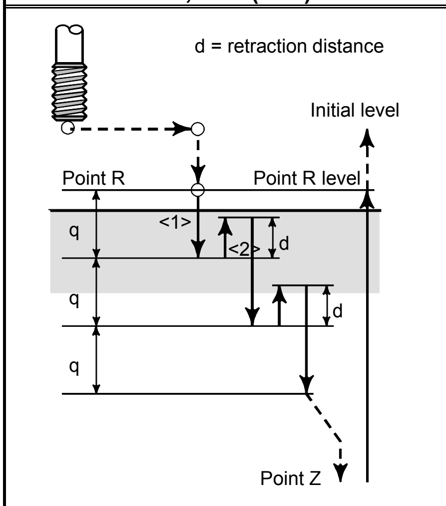
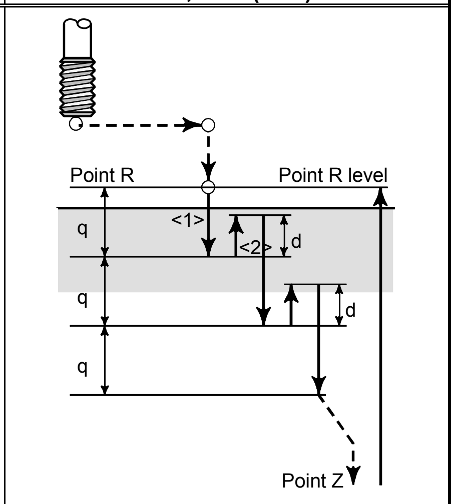
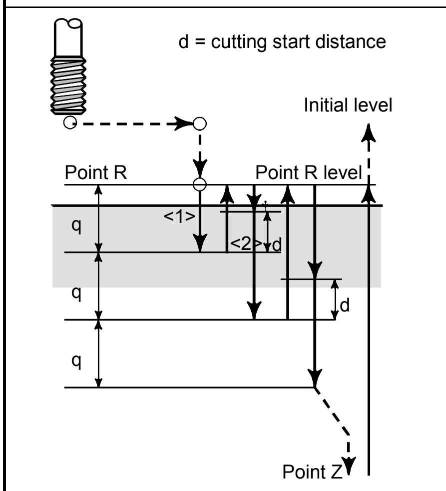
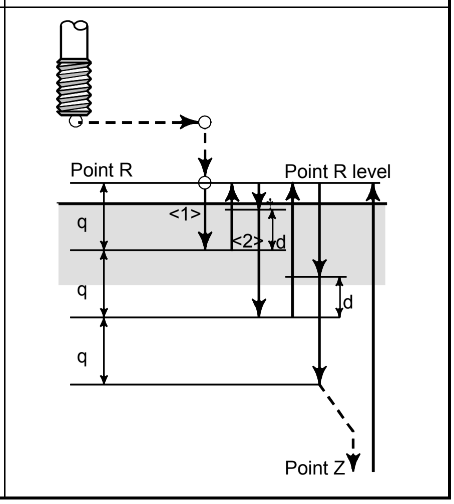
5.2.3 Peck Rigid Tapping Cycle (G84 or G74)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.
In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High-speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the PCP bit (bit 5) of parameter 5200.

Format

G84 (or G74) X_ Y_ Z_ R_ P_ Q_ F_ K_ ;

- X_ Y_ : Hole position data
- Z_ : The distance from point R to the bottom of the hole and the position of the bottom of the hole
- R_ : The distance from the initial level to point R level
- P_ : Dwell time at the bottom of the hole and at point R when a return is made
- Q_ : Depth of cut for each cutting feed
- F_ : The cutting feedrate
- K_ : Number of repeats (if required)

G84, G74 (G98)	G84, G74 (G99)
	
	

- High-speed peck tapping cycle (Parameter PCP(No.5200#5)=0)
  - <1> The tool operates at a normal cutting feedrate. The normal time constant is used.
  - <2> Retraction can be overridden. The retraction time constant is used.
- Peck tapping cycle (Parameter PCP(No.5200#5)=1)
  - <1> The tool operates at a normal cutting feedrate. The normal time constant is used.
  - <2> Retraction can be overridden. The retraction time constant is used.
  - <3> Retraction can be overridden. The normal time constant is used.

Explanation

- High-speed peck tapping cycle

After positioning along the X- and Y-axes, rapid traverse is performed to point R. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d. The DOV bit (bit 4) of parameter 5200 specifies whether retraction can be overridden or not. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.
Set the retraction distance, d, in parameter 5213.

- Peck tapping cycle

After positioning along the X- and Y-axes, rapid traverse is performed to point R level. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then a return is performed to point R. The DOV bit (bit 4) of parameter 5200 specifies whether the retraction can be overridden or not. The moving of cutting feedrate F is performed from point R to a position distance d from the end point of the last cutting, which is where cutting is restarted. For this moving of cutting feedrate F, the specification of the DOV bit (bit 4) of parameter 5200 is also valid. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.
Set d (distance to the point at which cutting is started) in parameter 5213.

- Acceleration/deceleration after interpolation

Linear or bell-shaped acceleration/deceleration can be applied.

- Look-ahead acceleration/deceleration before interpolation

Look-ahead acceleration/deceleration before interpolation is invalid.

- Override

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:

- Extraction override
- Override signal

Details are given later.

- Dry run

Dry run can be executed also in G84 (G74). When dry run is executed at the feedrate for the drilling axis in G84 (G74), tapping is performed according to the feedrate. Note that the spindle speed becomes faster at a higher dry run feedrate.

- Machine lock

Machine lock can be executed also in G84 (G74).
When G84 (G74) is executed in the machine lock state, the tool does not move along the drilling axis. Therefore, the spindle does not also rotate.

- Reset

When a reset is performed during rigid tapping, the rigid tapping mode is canceled and the spindle motor enters the normal mode. Note that the G84 (G74) mode is not canceled in this case when bit 6 (CLR) of parameter No. 3402 is set.

- Interlock

Interlock can also be applied in G84 (G74).

- Feed hold and single block

When bit 6 (FHD) of parameter No. 5200 is set to 0, feed hold and single block are invalid in the G84 (G74) mode. When this bit is set to 1, they are valid.

- Manual feed

For rigid tapping by manual handle feed, see the section "Rigid Tapping by Manual Handle."
With other manual operations, rigid tapping cannot be performed.

- Backlash compensation

In the rigid tapping mode, backlash compensation is applied to compensate the lost motion when the spindle rotates clockwise or counterclockwise. Set the amount of backlash in parameters Nos. 5321 to 5324.
Along the drilling axis, backlash compensation has been applied.

Limitation

- Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm PS0206 is issued.

- S command

- Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm PS0200.
- When the rigid tapping canned cycle is cancelled, the S command used for rigid tapping is cleared to S0.

- Distribution amount for the spindle

The maximum distribution amount is as follows (displayed on diagnostic screen No. 451):

- For a serial spindle: 32,767 pulses per 8 ms

This amount is changed according to the gear ratio setting for the position coder or rigid tapping command. If a setting is made to exceed the upper limit, alarm PS0202 is issued.

- F command

Specifying a value that exceeds the upper limit of cutting feedrate causes alarm PS0011.

- Unit of F command

	Metric input	Inch input	Remarks
G94	1 mm/min	0.01 inch/min	Decimal point programming allowed
G95	0.01 mm/rev	0.0001 inch/rev	Decimal point programming allowed

- M29

Specifying an S command or axis movement between M29 and G84 causes alarm PS0203.
Then, specifying M29 in the tapping cycle causes alarm PS0204.

- P/Q

Specify P and Q in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data.
When Q0 is specified, the peck rigid tapping cycle is not performed.

- Cancel

Do not specify a group 01 G code (G00 to G03 or G60 (when the MDL bit (bit 0 of parameter 5431) is set to 1)) and G84 in the same block. If they are specified together, G84 is canceled.

- Tool offset

In the canned cycle mode, tool offsets are ignored.

- Subprogram call

In the canned cycle mode, specify the subprogram call command M98P_ in an independent block.

5.2.4 Canned Cycle Cancel (G80)

The rigid tapping canned cycle is canceled. For how to cancel this cycle, see II-5.1.13.

> **NOTE**
> When the rigid tapping canned cycle is cancelled, the S value used for rigid tapping is also cleared (as if S0 is specified).
> Accordingly, the S command specified for rigid tapping cannot be used in a subsequent part of the program after the cancellation of the rigid tapping canned cycle.
> After canceling the rigid tapping canned cycle, specify a new S command as required.

5.2.5 Override during Rigid Tapping

Various types of override functions are invalid. The following override functions can be enabled by setting corresponding parameters:

- Extraction override
- Override signal

5.2.5.1 Extraction override

For extraction override, the fixed override set in the parameter or override specified in a program can be enabled at extraction (including retraction during peck drilling/high-speed peck drilling).

Explanation

- Specifying the override in the parameter

Set bit 4 (DOV) of parameter No. 5200 to 1 and set the override in parameter No. 5211.
An override from 0% to 200% in 1% steps can be set. Bit 3 (OVU) of parameter No. 5201 can be set to 1 to set an override from 0% to 2000% in 10% steps.

- Specifying the override in a program

Set bit 4 (DOV) of parameter No. 5200 and bit 4 (OV3) of parameter No. 5201 to 1. The spindle speed at extraction can be specified in the program.
Specify the spindle speed at extraction using address "J" in the block in which rigid tapping is specified.
Example) To specify 1000 min^{-1} for S at cutting and 2000 min^{-1} for S at extraction

```
  .
M29 S1000 ;
G84 Z-100. F1000. J2000 ;
  .
```

The difference in the spindle speed is converted to the actual override by the following calculation.
Therefore, the spindle speed at extraction may not be the same as that specified at address "J". If the override does not fall in the range between 100% and 200%, it is assumed to be 100%.

$$\text{Override (\%)} = \frac{\text{Spindle speed at extraction (specified at } J\text{)}}{\text{Spindle speed (specified at } S\text{)}} \times 100$$

The override to be applied is determined according to the setting of parameters and that in the command as shown in the table below.

Command \ Parameter setting		DOV = 1, OV3 = 1	DOV = 1, OV3 = 0	DOV = 0
Spindle speed at extraction specified at address "J"	Within the range between 100% to 200%	Command in the program	Parameter No. 5211	100%
	Outside the range between 100% to 200%	100%		
No spindle speed at extraction specified at address "J"		Parameter No. 5211		

NOTE

1 Do not use a decimal point in the value specified at address "J".
If a decimal point is used, the value is assumed as follows:
Example) When the increment system for the reference axis is IS-B
- When pocket calculator type decimal point programming is not used
The specified value is converted to the value for which the least input increment is considered.
"J200." is assumed to be 200000 min^{-1}.
- When pocket calculator type decimal point programming is used
The specified value is converted to the value obtained by rounding down to an integer.
"J200." is assumed to be 200 min^{-1}.

2 Do not use a minus sign in the value specified at address "J".
If a minus sign is used, a value outside the range between 100% to 200% is assumed.

3 The maximum override is obtained using the following equation so that the spindle speed to which override at extraction is applied do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5244). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

$$\text{Maximum override (\%)} = \frac{\text{Maximum spindle speed (specified in parameters)}}{\text{Spindle speed (specified at } S\text{)}} \times 100$$

4 When a value is specified at address "J" for specifying the spindle speed at extraction in the rigid tapping mode, it is valid until the canned cycle is canceled.

5.2.5.2 Override signal

By setting bit 4 (OVS) of parameter No. 5203 to 1, override can be applied to cutting/extraction operation during rigid tapping as follows:

- Applying override using the feedrate override signal
  (When the second feedrate override signal is enabled, the second feedrate override is applied to the feedrate to which feedrate override is applied.)
- Canceling override using the override cancel signal

There are the following relationships between this function and override to each operation:

- At cutting
  - When the override cancel signal is set to 0
    Value specified by the override signal
  - When the override cancel signal is set to 1
    100%
- At extraction
  - When the override cancel signal is set to 0
    Value specified by the override signal
  - When the override cancel signal is set to 1 and extraction override is disabled
    100%
  - When the override cancel signal is set to 1 and extraction override is enabled
    Value specified for extraction override

NOTE

1 The maximum override is obtained using the following equation so that the spindle speed to which override is applied do not exceed the maximum used gear speed (specified in parameters Nos. 5241 to 5244). For this reason, the obtained value is not the same as the maximum spindle speed depending on the override.

$$\text{Maximum override (\%)} = \frac{\text{Maximum spindle speed (specified in parameters)}}{\text{Spindle speed (specified at } S\text{)}} \times 100$$

2 Since override operation differs depending on the machine in use, refer to the manual provided by the machine tool builder.

5.3 OPTIONAL CHAMFERING AND CORNER R

Overview

Chamfering and corner R blocks can be inserted automatically between the following:

- Between linear interpolation and linear interpolation blocks
- Between linear interpolation and circular interpolation blocks
- Between circular interpolation and linear interpolation blocks
- Between circular interpolation and circular interpolation blocks

Format

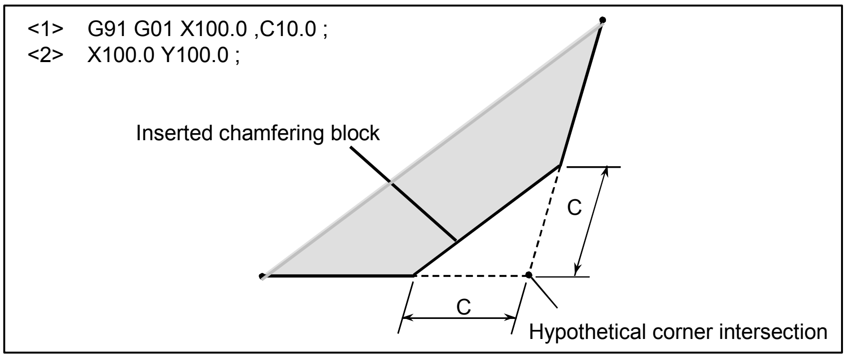
, C_	Chamfering
, R_	Corner R

Explanation

When the above specification is added to the end of a block that specifies linear interpolation (G01) or circular interpolation (G02 or G03), a chamfering or corner R block is inserted.
Blocks specifying chamfering and corner R can be specified consecutively.

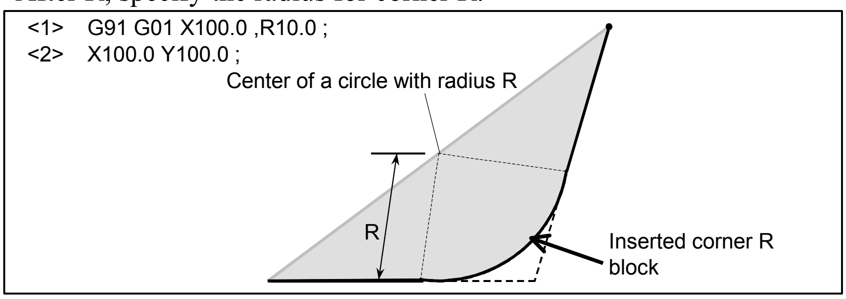
- Chamfering

After C, specify the distance from the hypothetical corner intersection to the start and end points. The hypothetical corner point is the corner point that would exist if chamfering were not performed.



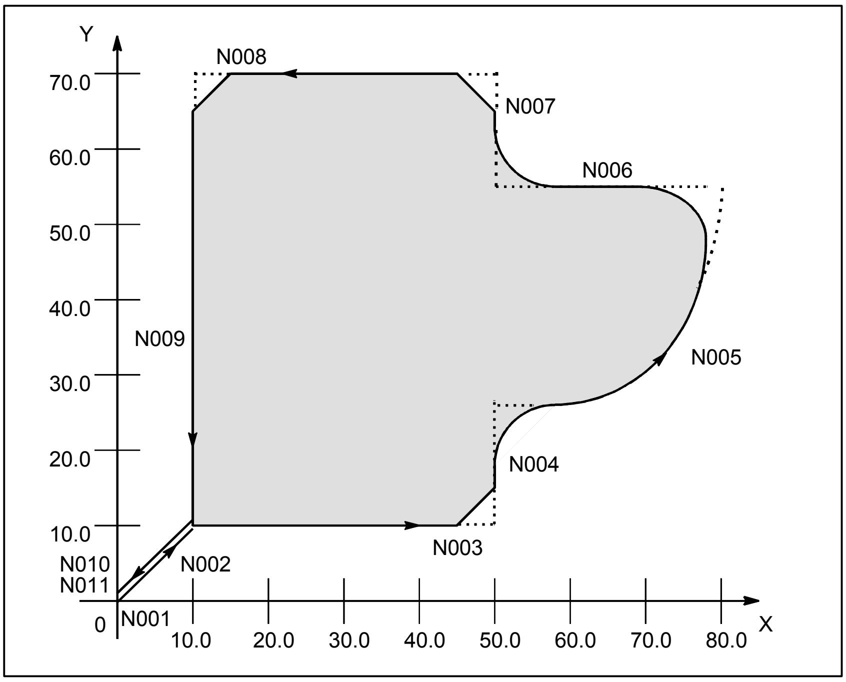
- Corner R

After R, specify the radius for corner R.



Example

```
N001 G92 G90 X0 Y0 ;
N002 G00 X10.0 Y10.0 ;
N003 G01 X50.0 F10.0 ,C5.0 ;
N004 Y25.0 ,R8.0 ;
N005 G03 X80.0 Y50.0 R30.0 ,R8.0 ;
N006 G01 X50.0 ,R8.0 ;
N007 Y70.0 ,C5.0 ;
N008 X10.0 ,C5.0 ;
N009 Y10.0 ;
N010 G00 X0 Y0 ;
N011 M0;
```



Limitation

- Invalid specification

Chamfering (,C) or corner R (,R) specified in a block other than a linear interpolation (G01) or circular interpolation (G02 or G03) block is ignored.

- Next block

A block specifying chamfering or corner R must be followed by a block that specifies a move command using linear interpolation (G01) or circular interpolation (G02 or G03). If the next block does not contain these specifications, alarm PS0051 is issued.
Between these blocks, however, only one block specifying G04 (dwell) can be inserted. The dwell is executed after execution of the inserted chamfering or corner R block.

- Exceeding the move range

If the inserted chamfering or corner R block causes the tool to go beyond the original interpolation move range, alarm PS0055 is issued.

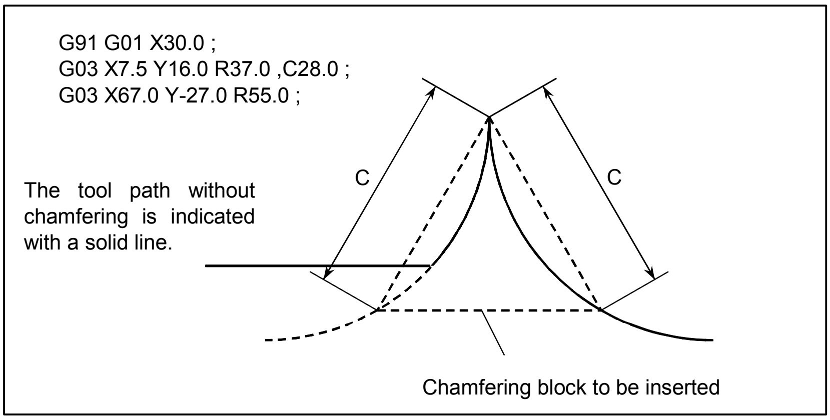


Fig 5.3 (a) Exceeding the move range

- Plane selection

A chamfering or corner R block is inserted only for a command to move the tool within the same plane.
Example:

When the U-axis is set as an axis parallel to the basic X-axis (by setting parameter No. 1022 to 5), the following program performs chamfering between cutting feed along the U-axis and that along the Y-axis:

```
G17 U0 Y0
G00 U100.0 Y100.0
G01 U200.0 F100, C30.0
Y200.0
```

The following program causes alarm PS0055, however. (Because chamfering is specified in the block to move the tool along the X-axis, which is not on the selected plane)

```
G17 U0 Y0
G00 U100.0 Y100.0
G01 X200.0 F100, C30.0
Y200.0
```

The following program also causes alarm PS0055. (Because the block next to the chamfering command moves the tool along the X-axis, which is not on the selected plane)

```
G17 U0 Y0
G00 U100.0 Y100.0
G01 Y200.0 F100, C30.0
X200.0
```

If a plane selection command (G17, G18, or G19) is specified in the block next to the block in which chamfering or corner R is specified, alarm PS0051 is issued.

- Travel distance 0

When two linear interpolation operations are performed, the chamfering or corner R block is regarded as having a travel distance of zero if the angle between the two straight lines is within ±1°. When linear interpolation and circular interpolation operations are performed, the corner R block is regarded as having a travel distance of zero if the angle between the straight line and the tangent to the arc at the intersection is within ±1°. When two circular interpolation operations are performed, the corner R block is regarded as having a travel distance of zero if the angle between the tangents to the arcs at the intersection is within ±1°.

- Single block operation

When the block in which chamfering or corner R is specified is executed in the single block mode, operation continues to the end point of the inserted chamfering or corner R block and the machine stops in the feed hold mode at the end point. When bit 0 (SBC) of parameter No. 5105 is set to 1, the machine stops in the feed hold mode also at the start point of the inserted chamfering or corner R block.

NOTE

1 When ",C" and ",R" are specified in the same block, the address specified last is valid.
2 If ",C" or ",R" is specified in a thread cutting command block, alarm PS0050 is issued.

5.4 INDEX TABLE INDEXING FUNCTION

By specifying indexing positions (angles) for the indexing axis (one rotation axis, A, B, or C), the index table of the machining center can be indexed.
Before and after indexing, the index table is automatically unclamped or clamped .

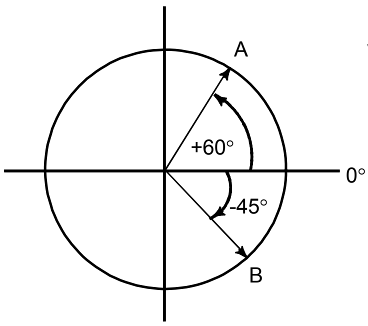
Explanation

- Indexing position

Specify an indexing position with address A, B, or C (set to bit 0 of parameter ROTx No.1006).
The indexing position is specified by either of the following (depending on bit 4 of parameter G90 No.5500):

1. Absolute value only
2. Absolute or incremental value depending on the specified G code: G90 or G91

A positive value indicates an indexing position in the counterclockwise direction. A negative value indicates an indexing position in the clockwise direction.
The minimum indexing angle of the index table is the value set to parameter 5512. Only multiples of the least input increment can be specified as the indexing angle. If any value that is not a multiple is specified, an alarm PS0135 occurs. Decimal fractions can also be entered. When a decimal fraction is entered, the 1's digit corresponds to degree units.



Value specified for rotation from A to B
(case 2 described above)
G90 B-45.0 ; or
G91 B-105.0;

- Direction and value of rotation

The direction of rotation and angular displacement are determined by either of the following two methods. Refer to the manual written by the machine tool builder to find out which method is applied.

1. Using the auxiliary function specified in parameter No. 5511 (Address) (Indexing position) (Miscellaneous function); Rotation in the negative direction (Address) (Indexing position); Rotation in the positive direction (No auxiliary functions are specified.)
   An angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360° when bit 2 of parameter ABS No. 5500 specifies this option.
   For example, when G90 B400.0 (auxiliary function); is specified at a position of 0 , the table is rotated by 40° in the negative direction.
2. Using no auxiliary functions
   By setting to bits 2, 3, and 4 of parameter ABS, INC,G90 No.5500, operation can be selected from the following two options.
   Select the operation by referring to the manual written by the machine tool builder.
   (1) Rotating in the direction in which an angular displacement becomes shortest
   This is valid only in absolute programming. A specified angular dis-placement greater than 360° is rounded down to the correspond-ing angular displacement within 360° when bit 2 of parameter ABS No.5500 specifies this option.
   For example, when G90 B400.0; is specified at a position of 0, the table is rotated by 40° in the positive direction.
   (2) Rotating in the specified direction
   In the absolute programming, the value set in bit 2 of parameter ABS No.5500 determines whether an angular displacement greater than 360° is rounded down to the corresponding angular displacement within 360°.
   In the incremental programming, the angular displacement is not rounded down. For example, when G90 B720.0; is specified at a position of 0, the table is rotated twice in the positive direction, when the angular displacement is not rounded down.

- Feedrate

The table is always rotated around the indexing axis in the rapid traverse mode.
Dry runs cannot be executed for the indexing axis.

> ⚠ **WARNING**
> If a reset is made during indexing of the index table, a reference position return must be made before each time the index table is indexed subsequently.

NOTE

1. Specify the indexing command in a single block. If the command is specified in a block in which another controlled axis is specified, alarm PS0136 occurs.
2. The waiting state which waits for completion of clamping or unclamping of the index table is indicated on diagnosis screen 12.
3. The auxiliary function specifying a negative direction is processed in the CNC. The relevant M code signal and completion signal are sent between the CNC and the machine.
4. If a reset is made while waiting for completion of clamping or unclamping, the clamp or unclamp signal is cleared and the CNC exits the completion wait state.

- Indexing function and other functions

Table 5.4 (a) Index indexing function and other functions

Item	Explanation
Relative position display	This value is rounded down when bit 1 of parameter REL No. 5500 specifies this option.
Absolute position display	This value is rounded down when bit 2 of parameterABS No. 5500 specifies this option.
Automatic return from the reference position (G29) 2nd reference position return (G30)	Impossible to return
Movement in the machine coordinate system (G53)	Impossible to move
Single direction positioning	Impossible to specify
2nd auxiliary function (B code)	Possible with any address other than B that of the indexing axis.
Operations while moving the indexing axis	Unless otherwise processed by the machine, feed hold, interlock and emerrgency stop can be executed. Machine lock can be executed after indexing is completed.
SERVO OFF signal	Disabled The indexing axis is usually in the servo-off state.
Incremental commands for indexing the index table	The workpiece coordinate system and machine coordinate system must always agree with each other on the indexing axis (the workpiece zero point offset value is zero.).
Operations for indexing the index table	Manual operation is disabled in the JOG, INC, or HANDLE mode. A manual reference position return can be made. If the axis selection signal is set to zero during manual reference position return, movement is stopped and the clamp command is not executed.

6 COMPENSATION FUNCTION

6.1 TOOL LENGTH COMPENSATION SHIFT TYPES

Overview

A tool length compensation operation can be performed by shifting the program coordinate system: The coordinate system containing the axis subject to tool length compensation is shifted by the tool length compensation value. A tool length compensation shift type can be selected with parameter TOS (parameter No. 5006#6). If no move command is specified together with the G43, G44, or G49 command, the tool will not move along the axis. If a move command is specified together with the G43, G44, or G49 command, the coordinate system will be shifted first, then the tool will move along the axis.

One of the following three methods is available, depending on the type of axis that can be subject to tool length compensation:

- Tool length compensation A
  Compensates the value of the tool length on the Z axis.
- Tool length compensation B
  Compensates the value of the tool length on one of the X, Y, and Z axis.
- Tool length compensation C
  Compensates the value of the tool length on a specified axis.

Format

- Tool length compensation A

G43 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the + side.

G44 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the - side.

G43 (or G44) : + (or -) side offset at which to start tool length compensation
H_ : Address specifying the tool length compensation value

- Tool length compensation B

G17 G43 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the + side.

G17 G44 Z_H_;
Shifts the coordinate system along the Z axis by the compensation value, to the - side.

G18 G43 Y_H_;
Shifts the coordinate system along the X axis by the compensation value, to the + side.

G18 G44 Y_H_;
Shifts the coordinate system along the X axis by the compensation value, to the - side.

G19 G43 X_H_;
Shifts the coordinate system along the Y axis by the compensation value, to the + side.

G19 G44 X_H_;
Shifts the coordinate system along the Y axis by the compensation value, to the - side.

G17 (or G18, G19) : Plane selection
G43 (or G44) : + (or -) side offset at which to start tool length compensation
H_ : Address specifying the tool length compensation value

- Tool length compensation C

G43 α_H_;
Shifts the coordinate system along a specified axis by the compensation value, to the + side.

G44 α_H_;
Shifts the coordinate system along a specified axis by the compensation value, to the - side.

G43 (or G44) : + (or -) side offset at which to start tool length compensation
α_ : Address of any one axis
H_ : Address specifying the tool length compensation value

- Tool length compensation cancel

G49; or H0; Tool length compensation cancel
G49 (or H0): Tool length compensation cancel

Explanation

- Offset direction

If the tool length compensation value specified with an H code (and stored in offset memory) is G43, the coordinate system is shifted to the + side; if G44, to the - side. If the sign of the tool length compensation value is -, the coordinate system is shifted to the - side if G43 and to the + side if G44. G43 and G44 are modal G codes; they remain valid until another G code in the same group is used.

- Specifying a tool length compensation value

The tool length compensation value corresponding to the number (offset number) specified with an H code (and stored in offset memory) is used. The tool length compensation corresponding to the offset number 0 always means 0. It is not possible to set a tool length compensation value corresponding to H0.

- Compensation axis

Specify one of tool length compensation types A, B, and C, using parameters TLC and TLB (No. 5001#0, #1).

- Specifying offset on two or more axes

Tool length compensation B enables offset on two or more axes by specifying offset axes in multiple blocks.

To perform offset on X and Y axes

G19 G43 H_; Performs offset on the X axis.
G18 G43 H_; Performs offset on the Y axis.

Tool length compensation C suppresses the generation of an alarm even if offset is performed on two or more axes at the same time, by setting TAL (No. 5001#3) to 1.

- Tool length compensation cancel

To cancel offset, specify either G49 or H0. Canceling offset causes the shifting of the coordinate system to be undone. If no move command is specified at this time, the tool will not move along the axis.

⚠ **CAUTION**

1. Specifying tool length compensation (a shift type) first and then executing an incremental programming causes the tool length compensation value to be reflected in the coordinates only, not in the travel distance of the machine; executing an absolute programming causes the tool length compensation value to be reflected in both the movement of the machine and the coordinates.
2. If a programmable mirror image is effective, the tool length compensation is applied in the specified direction.
3. No scaling magnification is applied to the tool length compensation value.
4. No coordinate system rotation is applied to the tool length compensation value. Tool length compensation is effective in the direction in which the offset is applied.
5. The tool length compensation operation is independent of the cutter compensation offset operation.
6. Three-dimensional coordinate conversion is applied to tool length compensation. If tool length compensation is made effective to multiple axes, the tool length compensation must be canceled for one axis at a time.
7. With the WINDOW command, changing parameter TOS during automatic operation does not cause the tool length compensation type to be changed.
8. If offset has been performed on two or more axes with tool length compensation B, a G49 command causes the offset to be canceled on all axes; H0 causes the offset to be canceled only on the axis vertical to the specified plane.
9. If the tool length compensation value is changed by changing the offset number, this simply means that the value is replaced by a new tool length compensation value; it does not mean that a new tool length compensation value is added to the old tool length compensation.

⚠ **CAUTION**

10 If reference position return (G28, G30, or G30.1) has been specified, tool length compensation is canceled for the axis specified at the time of positioning on the reference point; however, tool length compensation is not canceled for an un-specified axis. If reference position return has been specified in the same block as that containing tool length compensation cancel (G49), tool length compensation is canceled for both the specified and un-specified axes at the time of positioning on the mid-point.

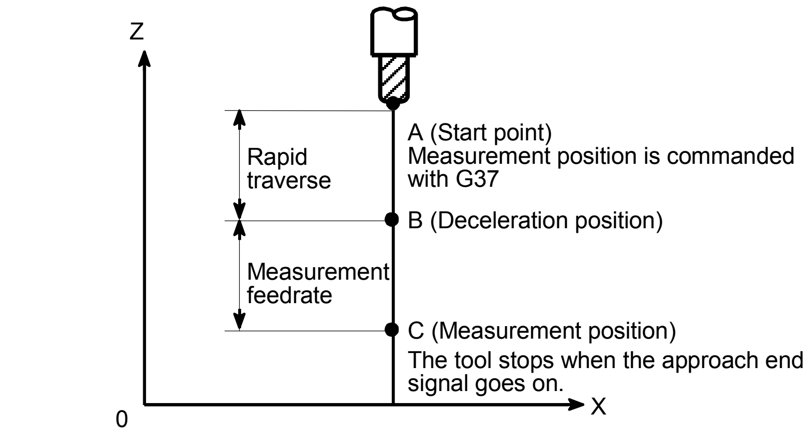
11 With a machine coordinate system command (G53), tool length compensation is canceled for the axis specified at the time of positioning on the specified point.

12 In high precision contour control mode, use tool length compensation of the axial movement type, with parameter TOS (No. 5006#6) being set to 0.

13 The tool length compensation vector canceled by specifying G53, G28, G30, or G30.1 during tool length compensation is restored as described below:
For tool length compensation types A and B, if parameter EVO (No. 5001#6) is 1, the vector is restored in the block buffered next; for all of tool length compensation types A, B, and C, it is restored in a block containing an H, G43, or G44 command if parameter is 0.

6.2 AUTOMATIC TOOL LENGTH MEASUREMENT (G37)

By issuing G37 the tool starts moving to the measurement position and keeps on moving till the approach end signal from the measurement device is output. Movement of the tool is stopped when the tool nose reaches the measurement position.
Difference between coordinate value when tool reaches the measurement position and coordinate value commanded by G37 is added to the tool length compensation amount currently used.



Compensation value = (Current compensation value) + [(Coordinates of the point at which the tool is stopped) - (Coordinates of the programmed measure ment position)]

Fig. 6.2 (a) Automatic tool length measurement

Format

G92 IP_ ;	Sets the workpiece coordinate system. (It can be set with G54 to G59. See Chapter "Coordinate System" in User's Manual (Common to T/M series.))
Hxx ;	Specifies an offset number for tool length compensation.
G90 G37 IP_ ;	Absolute programming

G37 is valid only in the block in which it is specified.
IP_ indicates the X-, Y-, Z-, or fourth axis.

Explanation

- Setting the workpiece coordinate system

Set the workpiece coordinate system so that a measurement can be made after moving the tool to the measurement position. The coordinate system must be the same as the workpiece coordinate system for programming.

- Specifying G37

Specify the absolute coordinates of the correct measurement position.
Execution of this command moves the tool at the rapid traverse rate toward the measurement position, reduces the federate halfway, then continuous to move it until the approach end signal from the measuring instrument is issued. When the tool nose reaches the measurement position, the measuring instrument sends an approach end signal to the CNC which stops the tool.

- Changing the offset value

The difference between the coordinates of the position at which the tool reaches for measurement and the coordinates specified by G37 is added to the current tool length compensation value.
Offset value =
(Current compensation value) + [(Coordinates of the position at which the tool reaches for measurement) - (Coordinates specified by G37)]
These offset values can be manually changed from MDI.

- Alarm

When automatic tool length measurement is executed, the tool moves as shown in Fig. 6.2 (b). If the approach end signal goes on while the tool is traveling from point B to point C, an alarm occurs. Unless the approach end signal goes on before the tool reaches point F, the same alarm occurs. The alarm number is PS0080.

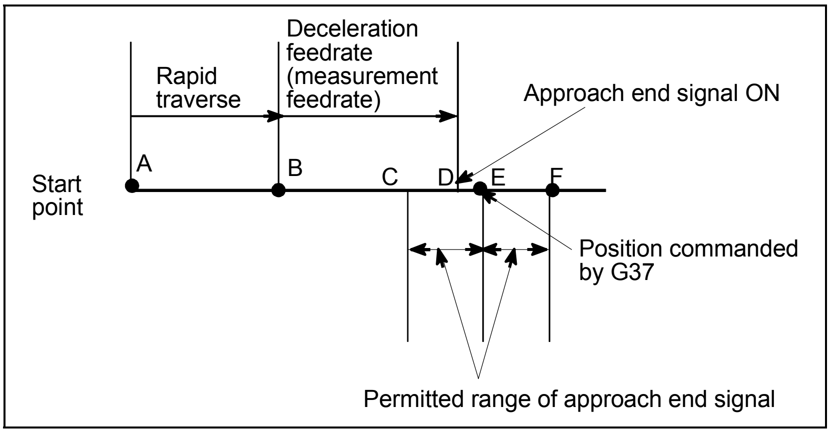


Fig. 6.2 (b) Tool movement to the measurement position

> ⚠ **WARNING**
> When a manual movement is inserted into a movement at a measurement federate, return the tool to the position before the inserted manual movement for restart.

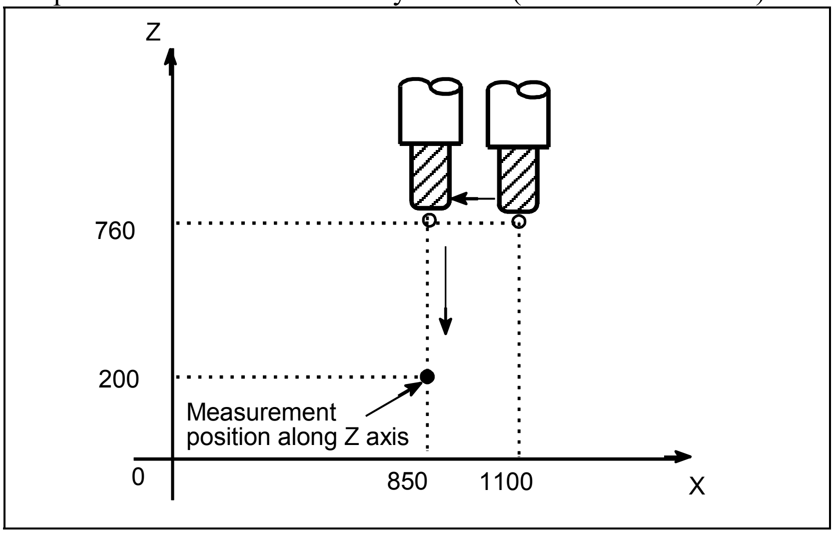
NOTE

1. When an H code is specified in the same block as G37, an alarm is generated. Specify H code before the block of G37.
2. The measurement speed (FP), γ, and ε are set as parameters (FP: No. 6241, γ: No. 6251, ε: No. 6254) by the machine tool builder. Make settings so that e are always positive and γ are always greater than ε.
3. When offset memory A is used, the offset value is changed. When offset memory B is used, the tool wear compensation value is changed. When offset memory C is used, the tool wear compensation value for the H code is changed.
4. A delay or variation in detection of the measurement position arrival signal is 0 to 2 msec on the CNC side excluding the PMC side (0.1 msec or less for high-speed measurement position arrival signal input (optional)). Therefore, the measurement error is the sum of 2 msec and a delay or variation (including a delay or variation on the receiver side) in propagation of the skip signal on the PMC side, multiplied by the feedrate set in parameter No. 6241.
5. A delay or variation in time after detection of the measurement position arrival signal until a feed stops is 0 to 8 msec. To calculate the amount of overrun, further consider a delay in acceleration/deceleration, servo delay, and delay on the PMC side.

Example

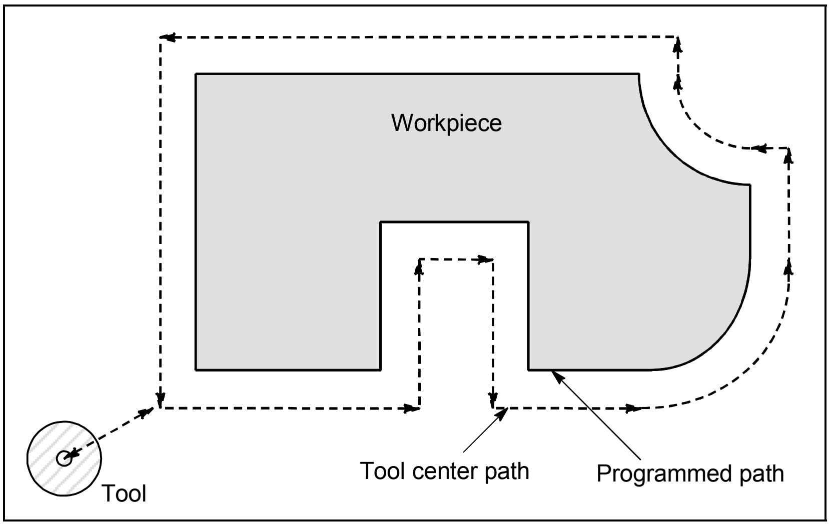
G92 Z760.0 X1100.0 ; Sets a workpiece coordinate system with respect to the programmed absolute zero point.
G00 G90 X850.0 ; Moves the tool to X850.0.
That is the tool is moved to a position that is a specified distance from the measurement position along the Z-axis.
H01 ; Specifies offset number 1.
G37 Z200.0 ; Moves the tool to the measurement position.
G00 Z204.0 ; Retracts the tool a small distance along the Z-axis.

For example, if the tool reaches the measurement position with Z198.0;, the compensation value must be corrected. Because the correct measurement position is at a distance of 200 mm, the compensation value is lessened by 2.0 mm (198.0 - 200.0 = -2.0).



6.3 TOOL OFFSET (G45 TO G48)

The programmed travel distance of the tool can be increased or decreased by a specified tool offset value or by twice the offset value. The tool offset function can also be applied to an additional axis.



Format

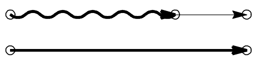
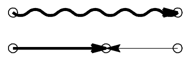
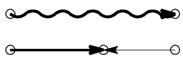
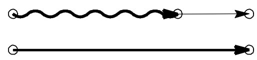
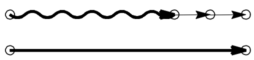
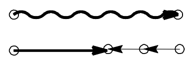
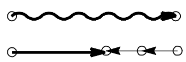
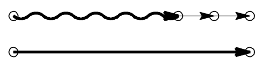
G45 IP_ D_ ;	Increase the travel distance by the tool offset value
G46 IP_ D_ ;	Decrease the travel distance by the tool offset value
G47 IP_ D_ ;	Increase the travel distance by twice the tool offset value
G48 IP_ D_ ;	Decrease the travel distance by twice the tool offset value
G45 to 48	: One-shot G code for increasing or decreasing the travel distance
IP_	: Command for moving the tool
D	Code for specifying the tool offset value

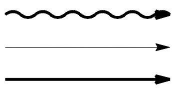
Explanation

- Increase and decrease

As shown in Table 6.3 (a), the travel distance of the tool is increased or decreased by the specified tool offset value.
In the absolute mode, the travel distance is increased or decreased as the tool is moved from the end point of the previous block to the position specified by the block containing G45 to G48.

Table 6.3 (a) Increase and decrease of the tool travel distance

G code	When a positive tool offset value is	When a negative tool offset value is specified
G45	Start point End point 	Start point End point 
G46	Start point End point 	Start point End point 
G47	Start point End point 	Start point End point 
G48	Start point End point 	Start point End point 



If a move command with a travel distance of zero is specified in the incremental programming (G91) mode, the tool is moved by the distance corresponding to the specified tool offset value.
If a move command with a travel distance of zero is specified in the absolute programming (G90) mode, the tool is not moved.

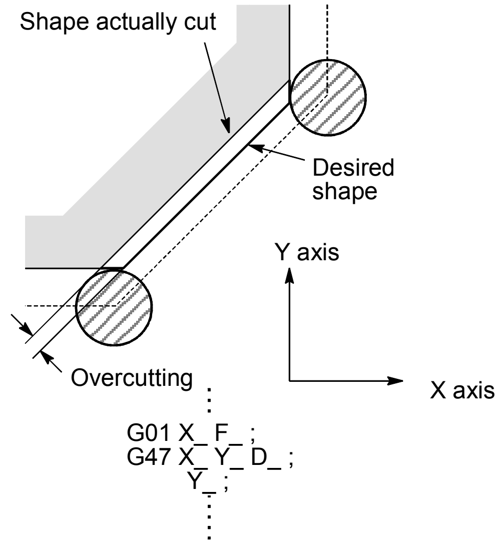
- Tool offset value

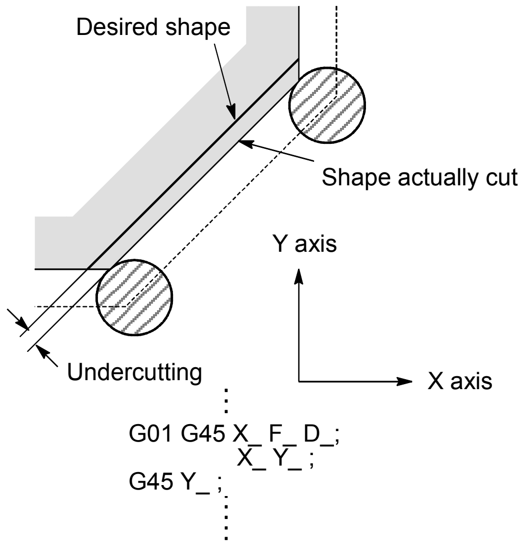
Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.
Tool offset values can be set within the following range:

D0 always indicates a tool offset value of zero.

⚠ CAUTION

1 When G45 to G48 is specified to n axes (n=1-6) simultaneously in a motion block, offset is applied to all n axes.
When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs.
Therefore, use cutter compensation (G40 or G42) shown in II-6.4 or 6.6.

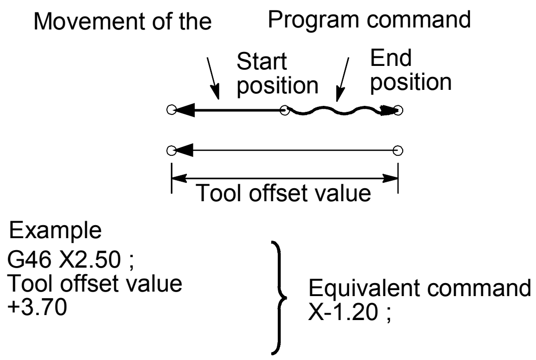




2 G45 to G48 (tool offset) must not be used in the G41 or G42 (cutter compensation) mode.

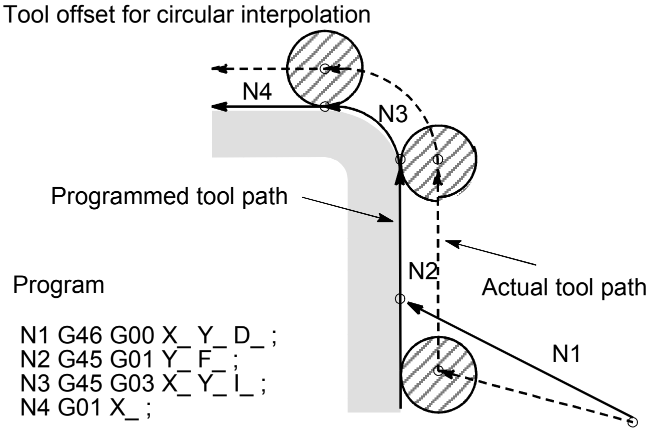
NOTE

1 When the specified direction is reversed by decrease as shown in the figure below, the tool moves in the opposite direction.



2 Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K by the parameter setting, providing that the coordinate system rotation be not specified at the same time. This function is provided for compatibility with the conventional CNC program without any cutter compensation. The function should not be used when a new CNC program is prepared.

Tool offset for circular interpolation

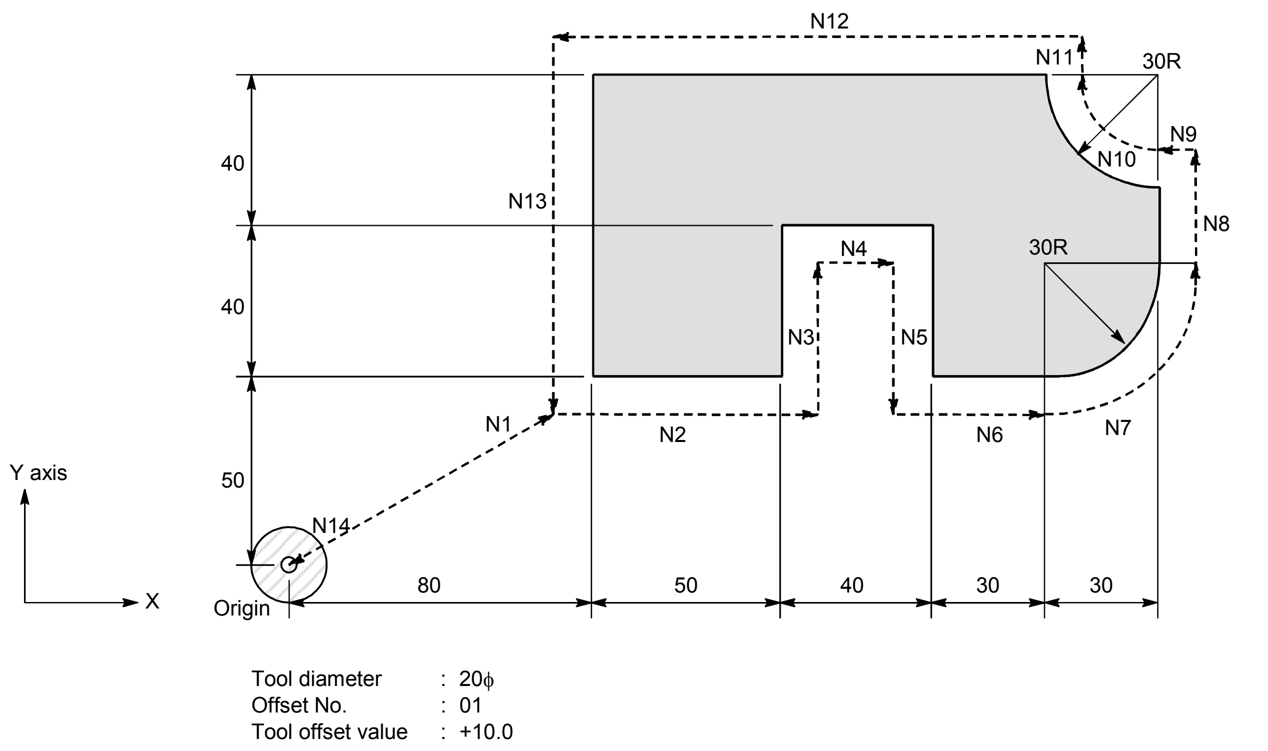


3 D code should be used in tool offset mode.

4 G45 to G48 are ignored in canned cycle mode. Perform tool offset by specifying G45 to G48 before entering canned cycle mode and cancel the offset after releasing the canned cycle mode.

Example

Program using tool offset



Program
N1 G91 G46 G00 X80.0 Y50.0 D01 ;
N2 G47 G01 X50.0 F120.0 ;
N3 Y40.0 ;
N4 G48 X40.0 ;
N5 Y-40.0 ;
N6 G45 X30.0 ;
N7 G45 G03 X30.0 Y30.0 J30.0 ;
N8 G45 G01 Y20.0 ;
N9 G46 X0 ; (Decreases toward the positive direction for movement amount "0". The tool moves in the -X direction by the offset value.)
N10 G46 G02 X-30.0 Y30.0 J30.0 ;
N11 G45 G01 Y0 ; (Increase toward the positive direction for movement amount "0". The tool moves in the +Y direction by the offset value.)
N12 G47 X-120.0 ;
N13 G47 Y-80.0 ;
N14 G46 G00 X80.0 Y-50.0 ;

6.4 OVERVIEW OF CUTTER COMPENSATION (G40-G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 6.4 (a)).
To make an offset as large as the radius of the tool, CNC first creates an offset vector with a length equal to the radius of the tool (start-up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head positions to the center of the tool.
If a linear interpolation or circular interpolation command is specified after start-up, the tool path can be shifted by the length of the offset vector during machining.
To return the tool to the start point at the end of machining, cancel the cutter compensation mode.

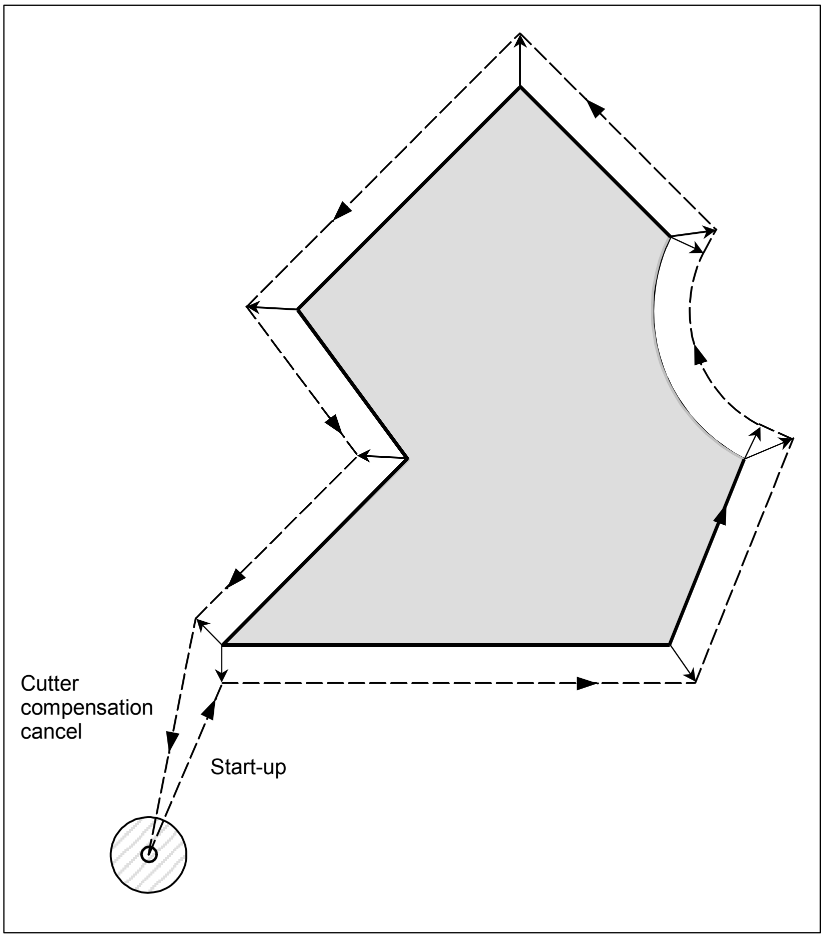


Fig. 6.4 (a) Outline of cutter compensation

Format

- Start up (cutter compensation start)

G00(or G01)G41(or G42) IP_D_;

G41 : Cutter compensation left (Group 07)
G42 : Cutter compensation right (Group 07)
IP_ : Command for axis movement
D_ : Code for specifying as the cutter compensation value (1-3 digits) (D code)

- Cutter compensation cancel (offset mode cancel)

G40 IP_;

G40 : Cutter compensation cancel (Group 07) (Offset mode cancel)
IP_ : Command for axis movement

- Selection of the offset plane

Offset plane	Command for plane selection	IP_
XpYp	G17 ;	Xp_Yp_
ZpXp	G18 ;	Xp_Zp_
YpZp	G19 ;	Yp_Zp_

Explanation

- Offset cancel mode

At the beginning when power is applied the control is in the cancel mode. In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path.

- Start-up

When a cutter compensation command (G41 or G42, D code other than 0) is specified in the offset cancel mode, the CNC enters the offset mode.
Moving the tool with this command is called start-up.
Specify positioning (G00) or linear interpolation (G01) for start-up.
If circular interpolation (G02, G03) or involute interpolation (G02.2, G03.2) is specified, alarm PS0034 occurs.
For the start-up and subsequent blocks, the CNC prereads as many blocks as the number of preread blocks set in the parameter (No. 19625).

- Offset mode

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03).
If three or more blocks that move the tool cannot be read in offset mode, the tool may make either an excessive or insufficient cut.
If the offset plane is switched in the offset mode, alarm PS0037 occurs and the tool is stopped.

- Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC enters the offset cancel mode, and the action of this block is called the offset cancel.

1. G40 has been commanded.
2. 0 has been commanded as the offset number for cutter compensation (D code).

When performing offset cancel, circular arc commands (G02 and G03) and involute commands (G02.2 and G03.2) are not available. If these commands are specified, an PS0034 is generated and the tool stops. In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer.
In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block.
Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for cutter compensation.

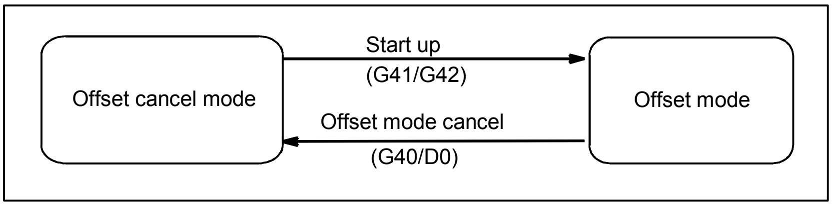


Fig. 6.4 (b) Changing the offset mode

- Change of the cutter compensation value

In general, the cutter compensation value shall be changed in the cancel mode, when changing tools. If the cutter compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new cutter compensation value.

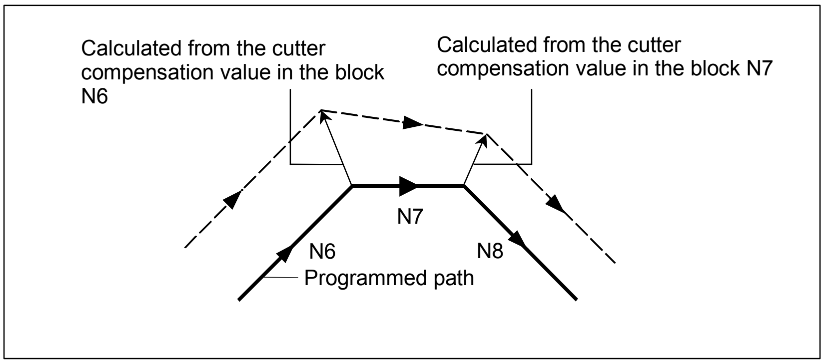


Fig. 6.4 (c) Changing the cutter compensation value

- Positive/negative cutter compensation value and tool center path

If the compensation value is negative (–), distribution is made for a figure in
which G41's and G42's are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa.
Fig. 6.4 (d) shows one example.
Generally, the compensation value is programmed to be positive (+).
When a tool path is programmed as in <1>, if the compensation value is made negative (–), the tool center moves as in <2>, and vice versa. Consequently, the same program permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the compensation value.
Applicable if start-up and cancel is A type. (See the descriptions about the start-up of cutter compensation.)

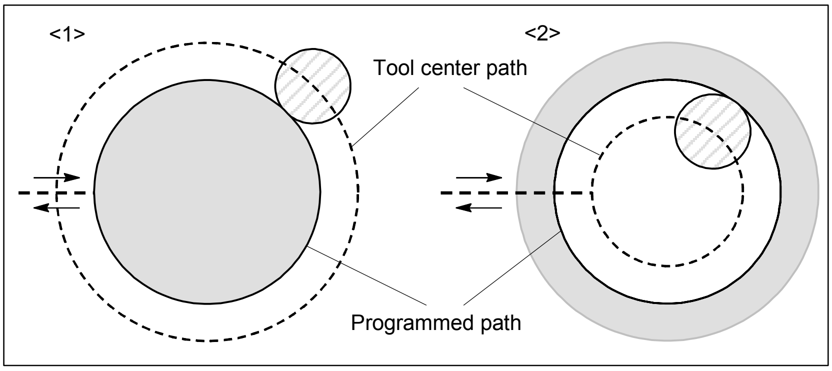


Fig. 6.4 (d) Tool center paths when positive and negative cutter compensation values are specified

- Cutter compensation value setting

Assign a cutter compensation values to the D codes on the MDI panel.

> **NOTE**
> The cutter compensation value for which the D code corresponds to 0 always means 0.
> It is not possible to set the cutter compensation value corresponding to D0.

- Valid compensation value range

The valid range of values that can be set as a compensation value is either of the following, depending on the parameters OFE, OFD, OFC, and OFA (No. 5042 #3 to #0).

Valid compensation range (metric input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±9999.99 mm
0	0	0	0	±9999.999 mm
0	0	1	0	±9999.9999 mm
0	1	0	0	±9999.99999 mm
1	0	0	0	±999.999999 mm

Valid compensation range (inch input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±999.999 inch
0	0	0	0	±999.9999 inch
0	0	1	0	±999.99999 inch
0	1	0	0	±999.999999 inch
1	0	0	0	±99.9999999 inch

The compensation value corresponding to offset No. 0 always means 0. It is not possible to set the compensation value corresponding to offset No. 0.

- Offset vector

The offset vector is the two dimensional vector that is equal to the cutter compensation value assigned by D code. It is calculated inside the control unit, and its direction is up-dated in accordance with the progress of the tool in each block.
The offset vector is deleted by reset.

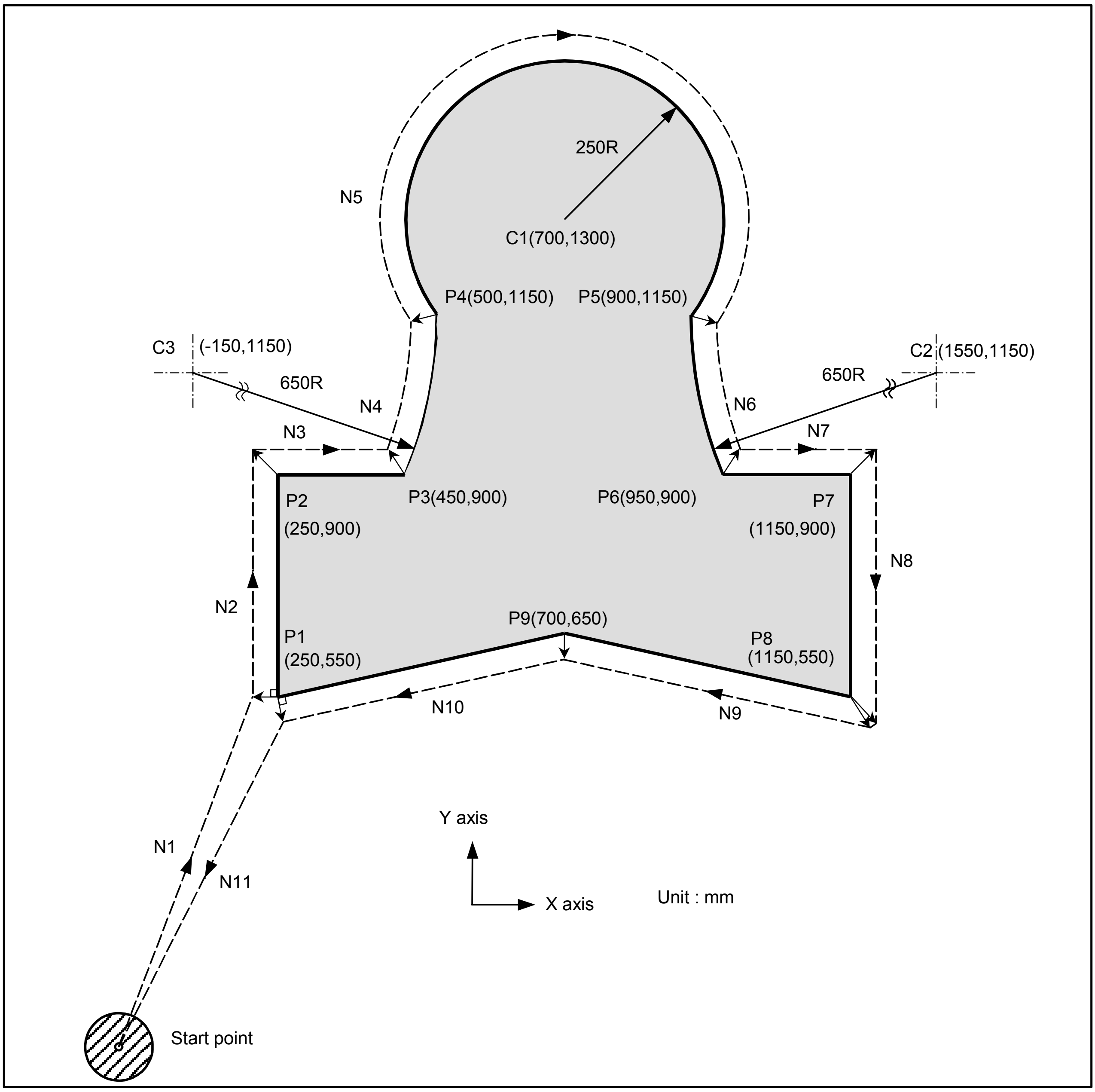
- Specifying a cutter compensation value

Specify a cutter compensation value with a number assigned to it. The number consists of 1 to 3 digits after address D (D code).
The D code is valid until another D code is specified. The D code is used to specify the tool offset value as well as the cutter compensation value.

- Plane selection and vector

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane.
Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are.
In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.
The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, an PS0037 is displayed and the machine is stopped.

Example


250R
N5
C1(700,1300)
P4(500,1150)
P5(900,1150)
C3
(-150,1150)
650R
N4
N3
C2
(1550,1150)
650R
N6
N7
P2
(250,900)
P3(450,900)
P6(950,900)
P7
(1150,900)
N8
N2
P9(700,650)
P1
(250,550)
P8
(1150,550)
N10
N9
Y axis
N1
N11
X axis
Unit : mm
Start point

	G92 X0 Y0 Z0 ;	Specifies absolute coordinates. The tool is positioned at the start point (X0, Y0, Z0).
N1	**G90 G17 G00 G41 D07 X250.0 Y550.0 ;**	Starts cutter compensation (start-up). The tool is shifted to the left of the programmed path by the distance specified in D07. In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15 mm).
N2	**G01 Y900.0 F150 ;**	Specifies machining from P1 to P2.
N3	**X450.0 ;**	Specifies machining from P2 to P3.
N4	**G03 X500.0 Y1150.0 R650.0 ;**	Specifies machining from P3 to P4.
N5	**G02 X900.0 R-250.0 ;**	Specifies machining from P4 to P5.
N6	**G03 X950.0 Y900.0 R650.0 ;**	Specifies machining from P5 to P6.
N7	**G01 X1150.0 ;**	Specifies machining from P6 to P7.
N8	**Y550.0 ;**	Specifies machining from P7 to P8.
N9	**X700.0 Y650.0 ;**	Specifies machining from P8 to P9.
N10	**X250.0 Y550.0 ;**	Specifies machining from P9 to P1.
N11	**G00 G40 X0 Y0 ;**	Cancels the offset mode. The tool is returned to the start point (X0, Y0, Z0).

6.5 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION (G40-G42)

The tool nose radius compensation function automatically compensates for the errors due to the tool nose roundness.

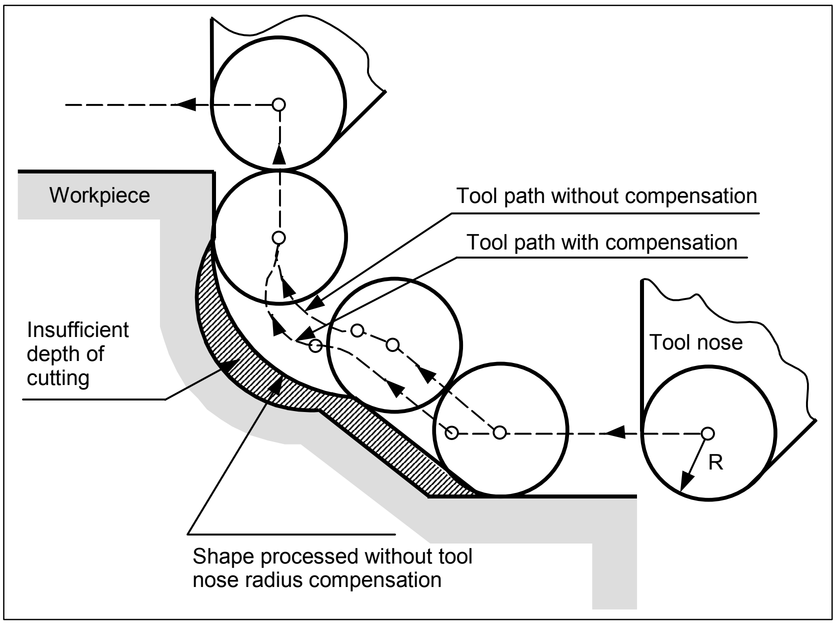


Fig 6.5 (a) Tool path of tool nose radius compensation

6.5.1 Imaginary Tool Nose

The tool nose at position A in Fig. 6.5.1 (a) does not actually exist.
The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start point than the imaginary tool nose.
Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.
The position relationship when the tool is set to the start point is shown in Fig. 6.5.1 (a).

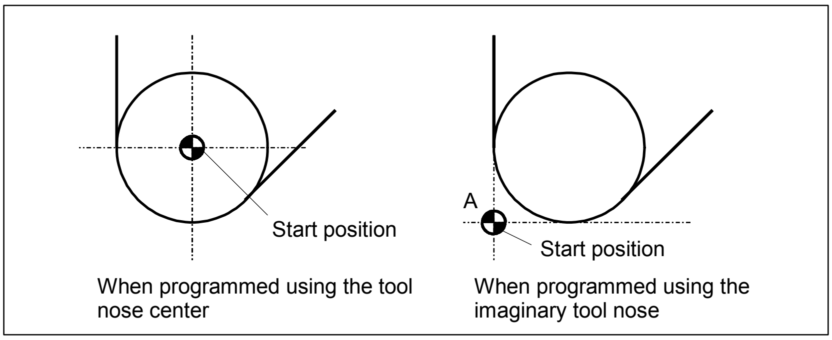


Fig. 6.5.1 (a) Tool nose radius center and imaginary tool nose

⚠ CAUTION

In a machine with reference positions, a standard position like the turret center can be placed over the start point. The distance from this standard position to the tool nose radius center or the imaginary tool nose is compensated by the tool length compensation function.

Setting the distance from the standard position to the tool nose radius center as the offset value is the same as placing the tool nose radius center over the start point, while setting the distance from the standard position to the imaginary tool nose is the same as placing the imaginary tool nose over the standard position. To set the offset value, it is usually easier to measure the distance from the standard position to the imaginary tool nose than from the standard position to the tool nose radius center.

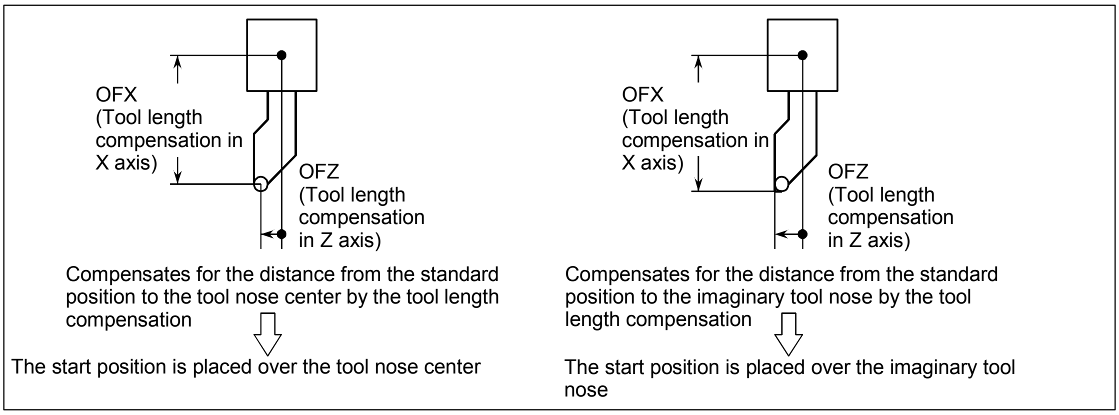


Fig. 6.5.1 (b) Tool length compensation when the turret center is placed over the start point

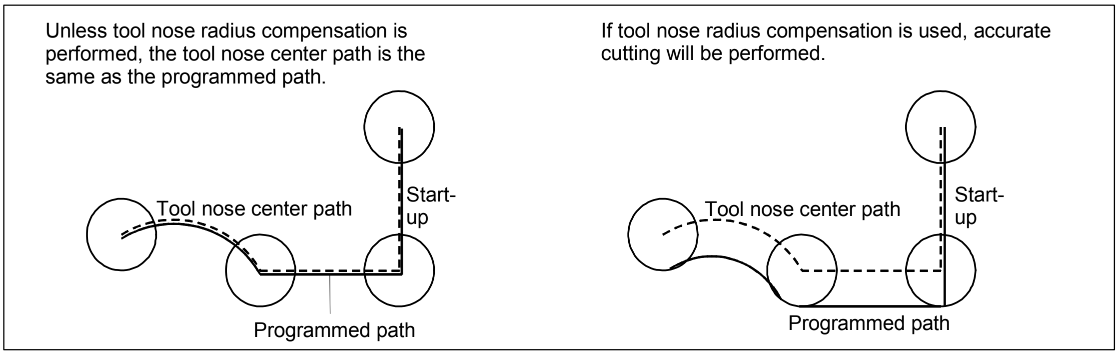


Fig. 6.5.1 (c) Tool path when programming using the tool nose center

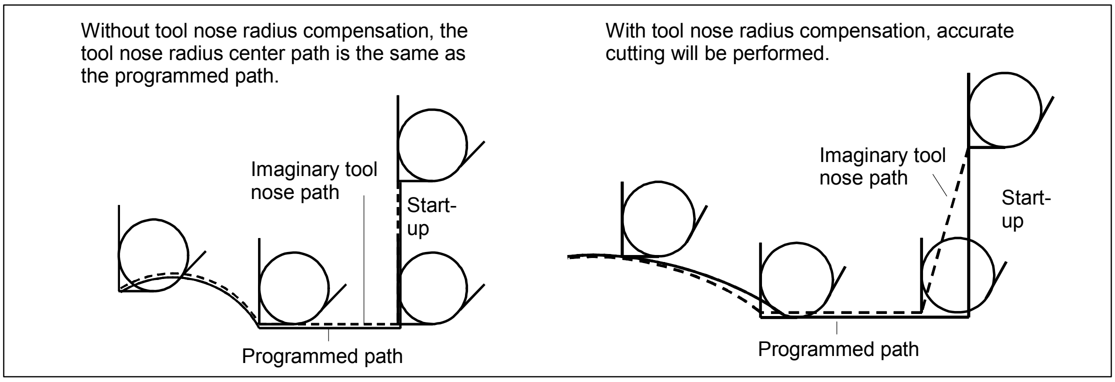


Fig. 6.5.1 (d) Tool path when programming using the imaginary tool nose

6.5.2 Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.
The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig. 6.5.2 (a) below together with their corresponding codes. This Fig 6.5.2 (a) illustrates the relation between the tool and the start point. The following apply when the tool geometry offset and tool wear offset option are selected.

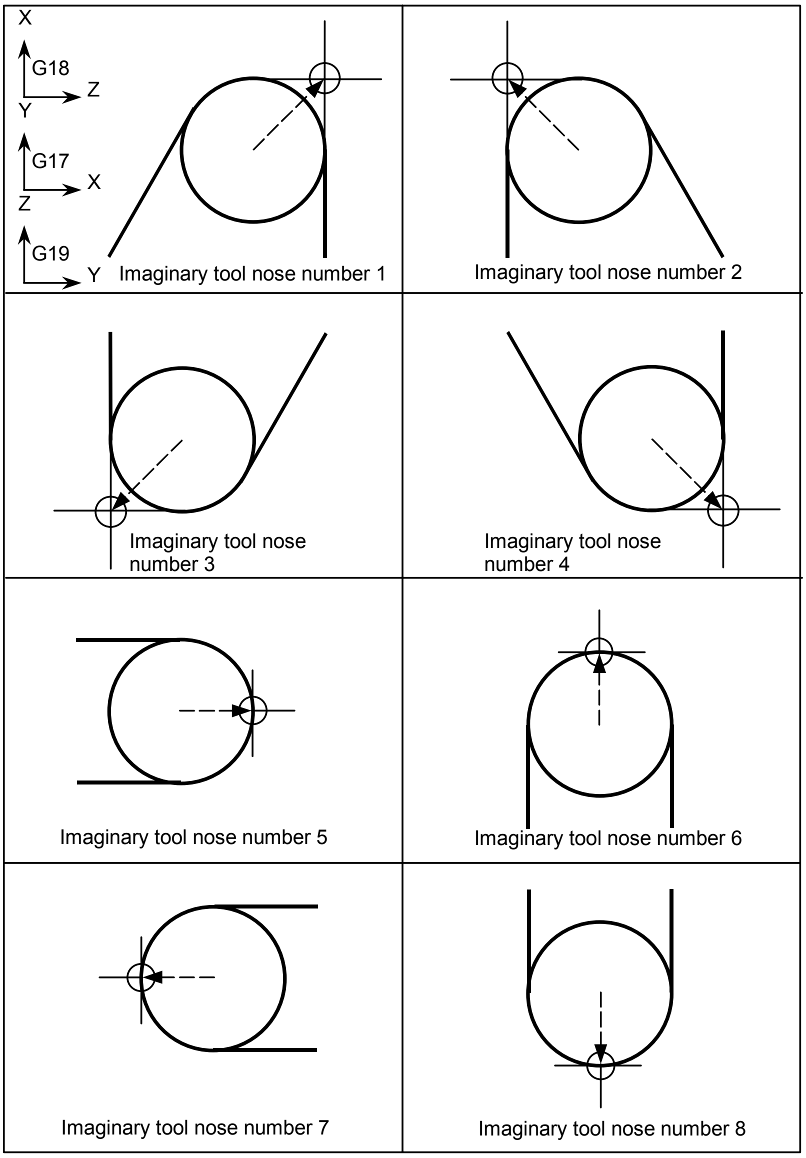
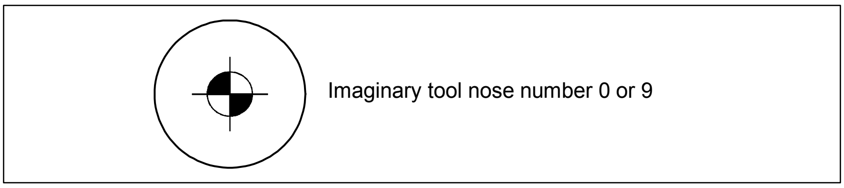


Fig. 6.5.2 (a) Direction of imaginary tool nose

Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start point. Set imaginary tool nose number to address OFT for each offset number.



6.5.3 Offset Number and Offset Value

Explanation

- Offset number and offset value

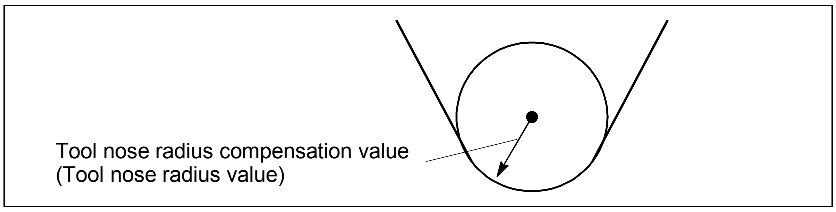


Table 6.5.3 (a) Offset number and offset value (example)

Offset number Up to 999 sets	(Tool compensation value)	(Direction of imaginary tool nose)
001	0.200	1
002	0.250	2
003	0.120	6
004	:	:
005	:	:
:	:	:

- Command of offset value

An offset number is specified with the D code.

- Setting range of offset value

The range of values that can be set as a compensation value is either of the following, depending on the parameters OFE, OFD, OFC, and OFA (No. 5042#3 to No. 5042#0).

Valid compensation range (metric input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±9999.99 mm
0	0	0	0	±9999.999 mm
0	0	1	0	±9999.9999 mm
0	1	0	0	±9999.99999 mm
1	0	0	0	±999.999999 mm

Valid compensation range (inch input)

OFE	OFD	OFC	OFA	Range
0	0	0	1	±999.999 inch
0	0	0	0	±999.9999 inch
0	0	1	0	±999.99999 inch
0	1	0	0	±999.999999 inch
1	0	0	0	±99.9999999 inch

The offset value corresponding to the offset number 0 is always 0.
No offset value can be set to offset number 0.

6.5.4 Workpiece Position and Move Command

In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.

G code	Workpiece position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

The tool is offset to the opposite side of the workpiece.

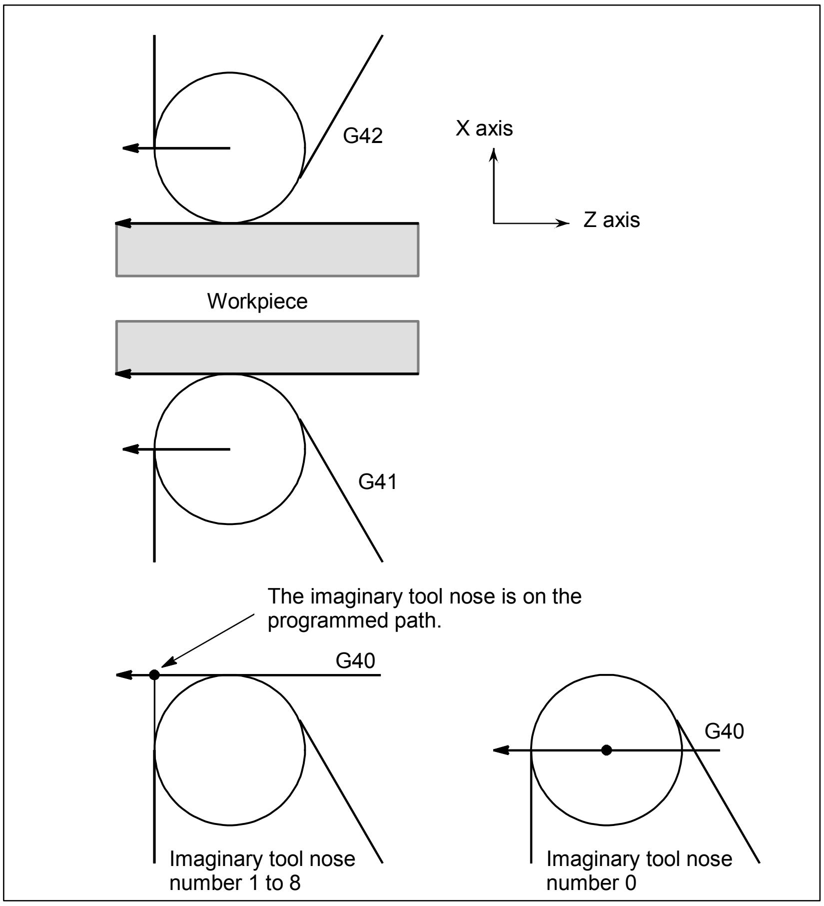


Fig. 6.5.4 (a) Workpiece position

The workpiece position can be changed by setting the coordinate system as shown below.

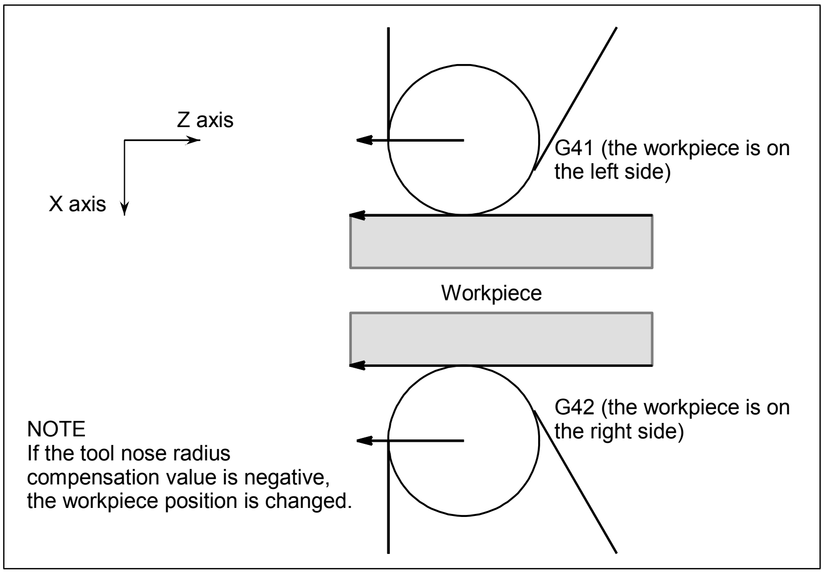


Fig. 6.5.4 (b) When the workpiece position is changed

G40, G41, and, G42 are modal.
Don't specify G41 while in the G41 mode. If you do, compensation will not work properly.
Don't specify G42 while in the G42 mode for the same reason.
G41 or G42 mode blocks in which G41 or G42 are not specified are expressed by (G41) or (G42) respectively.

> ⚠ **CAUTION**
> If the sign of the compensation value is changed from plus to minus and vice versa, the offset vector of tool nose radius compensation is reversed, but the direction of the imaginary tool nose does not change. For a use in which the imaginary tool nose is adjusted to the starting point, therefore, do not change the sign of the compensation value for the assumed program.

Explanation

- Tool movement when the workpiece position does not change

When the tool is moving, the tool nose maintains contact with the workpiece.

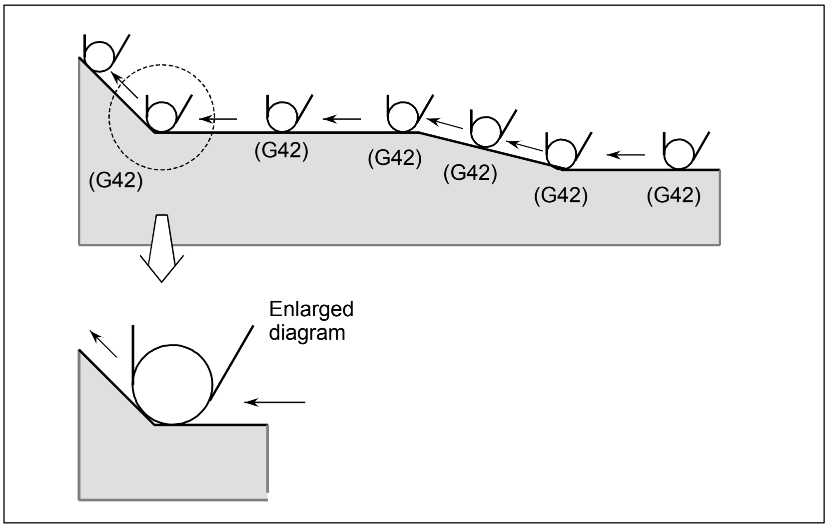


Fig. 6.5.4 (c) Tool movement when the workpiece position does not change

- Tool movement when the workpiece position changes

The workpiece position against the tool changes at the corner of the programmed path as shown in the following figure.

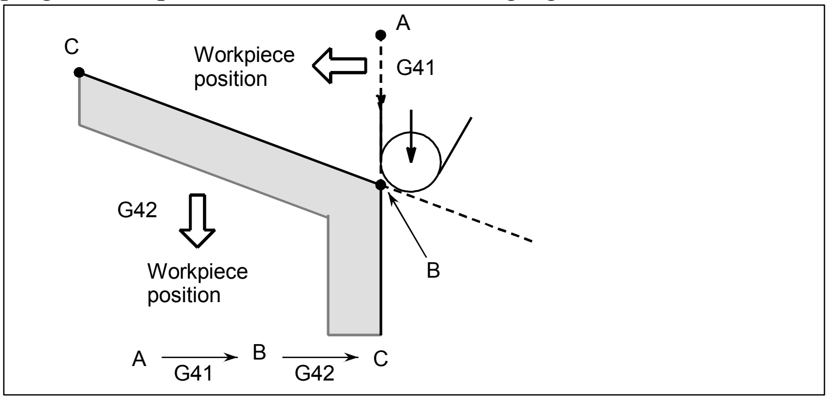


Fig. 6.5.4 (d) Tool movement when the workpiece position changes

Although the workpiece does not exist on the right side of the programmed path in the above case, the existence of the workpiece is assumed in the movement from A to B. The workpiece position must not be changed in the block next to the start-up block. In the above example, if the block specifying motion from A to B were the start-up block, the tool path would not be the same as the one shown.

- Start-up

The block in which the mode changes to G41 or G42 from G40 is called the start-up block.

G40 _ ;
G41 _ ; (Start-up block)

Transient tool movements for offset are performed in the start-up block. In the block after the start-up block, the tool nose center is positioned Vertically to the programmed path of that block at the start point.

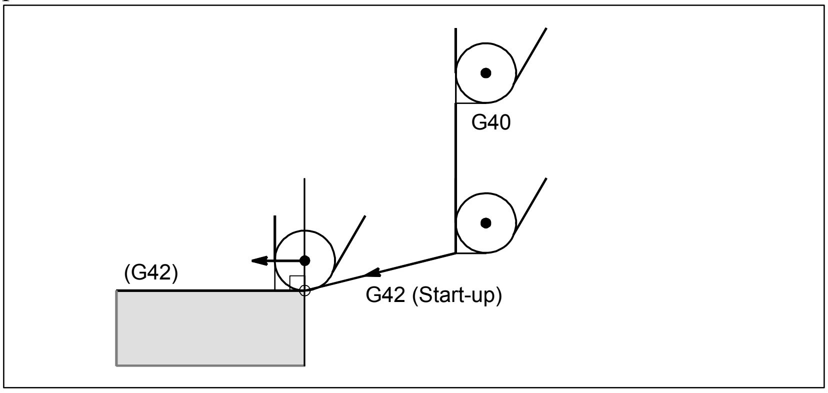


Fig. 6.5.4 (e) Start-up

- Offset cancel

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

G41 _ ;
G40 _ ; (Offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block.
The tool is positioned at the end point in the offset cancel block (G40) as shown below.

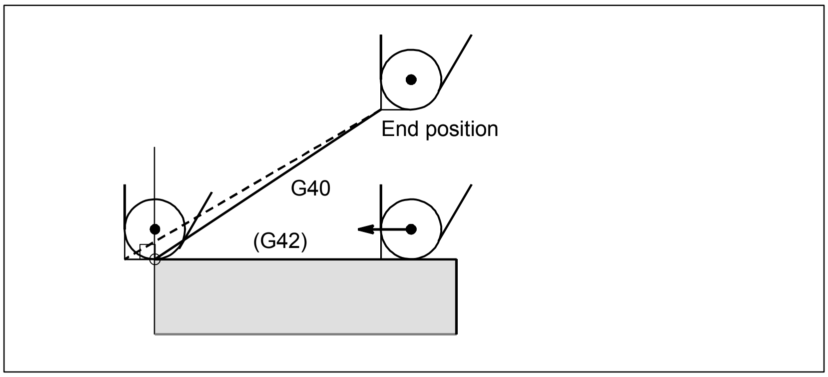


Fig. 6.5.4 (f) Offset cancel

- Changing the compensation value

In general, the compensation value is to be changed when the tool is changed in offset cancel mode. If the compensation value is changed in offset mode, however, the vector at the end point of the block is calculated using the compensation value specified in that same block.
The same applies if the imaginary tool nose direction and the tool offset value are changed.

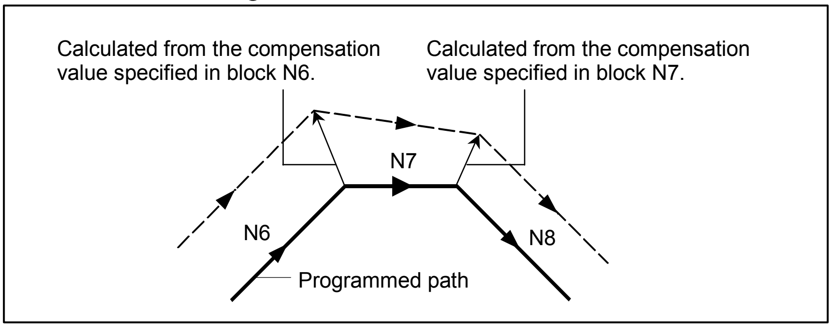


Fig. 6.5.4 (g) Changing the compensation value

- Specification of G41/G42 in G41/G42 mode

When a G41 or G42 code is specified again in G41/G42 mode, the tool nose center is positioned vertical to the programmed path of the preceding block at the end point of the preceding block.

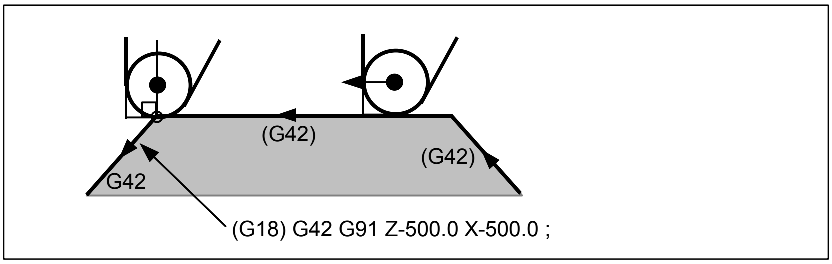


Fig. 6.5.4 (h) Specification of G41/G42 in G41/G42 mode

In the block that first changes from G40 to G41/G42, the above positioning of the tool nose center is not performed.

- Tool movement when the moving direction of the tool in a block which includes a G40 (offset cancel) command is different from the direction of the workpiece

When you wish to retract the tool in the direction specified by X and Z canceling the tool nose radius compensation at the end of machining the first block in Fig. 6.5.4 (i), specify the following :

G40 X _ Z _ I _ K _ ;

where I and K are the direction of the workpiece in the next block, which is specified in incremental mode.

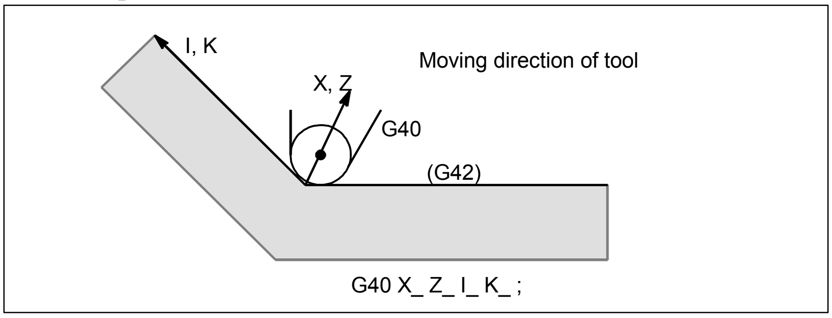


Fig. 6.5.4 (i) If I and K are specified in the same block as G40

Thus, this prevents the tool from overcutting, as shown in Fig. 6.5.4 (j).

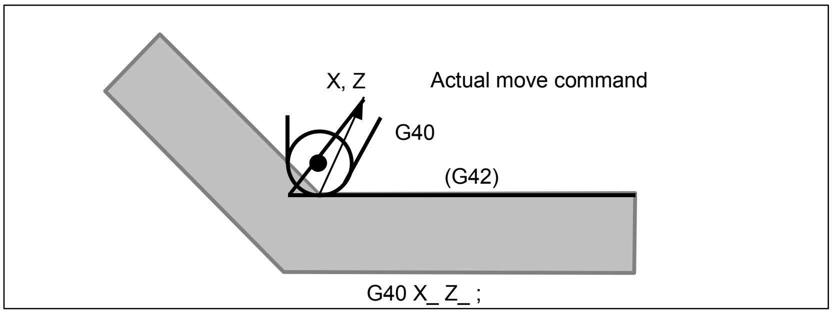


Fig. 6.5.4 (j) Case in which overcutting occurs in the same block as G40

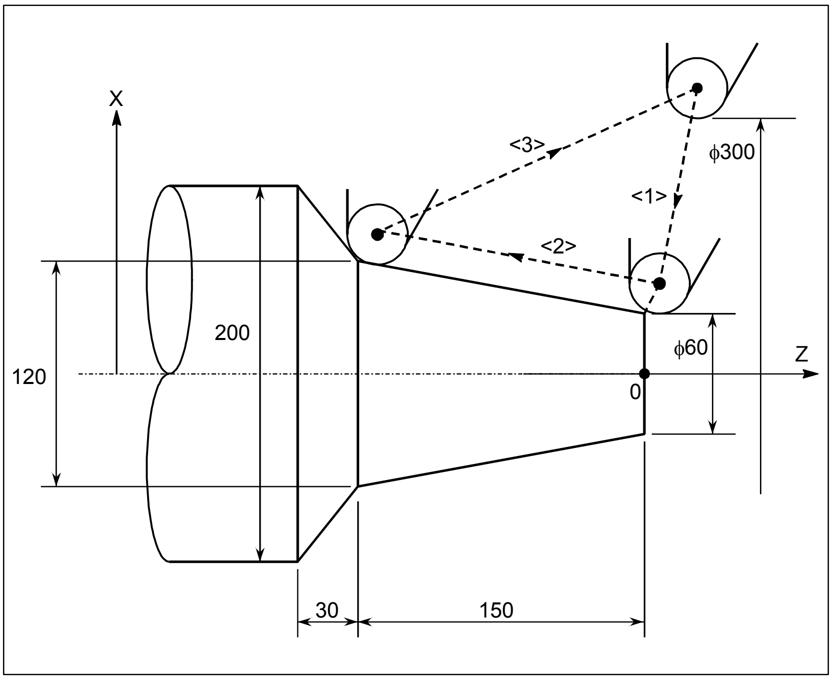
The workpiece position specified by addresses I and K is the same as that in the preceding block.
Specify I_K_; in the same block as G40. If it is specified in the same block as G02 or G03, it is assumed to be the center of the arc.

G40 X_ Z_ I_ K_ ;	Tool nose radius compensation
G02 X_ Z_ I_ K_ ;	Circular interpolation

If I and/or K is specified with G40 in the cancel mode, the I and/or K is ignored. The numeral is followed I and K should always be specified as radius values.

G40 G01 X_ Z_ ;
G40 G01 X_ Z_ I_ K_ ; Offset cancel mode (I and K are ineffective.)

Example



(G40 mode)

<1> **G42 G00 X60.0 ;**

<2> **G01 X120.0 Z-150.0 F10 ;**

<3> **G40 G00 X300.0 Z0 I40.0 K-30.0 ;**

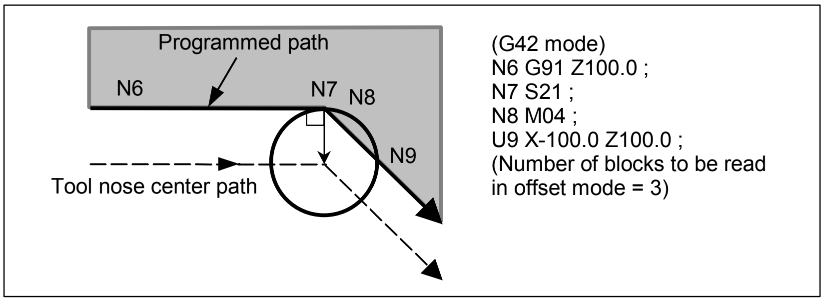
6.5.5 Notes on Tool Nose Radius Compensation

Explanation

- Blocks without a move command that are specified in offset mode

<1>	**M05 ;**	M code output
<2>	**S210 ;**	S code output
<3>	**G04 X10.0 ;**	Dwell
<4>	**G22 X100000 ;**	Machining area setting
<5>	**G91 G01 X0 ;**	Feed distance of zero
<6>	**G90 ;**	G code only
<7>	**G10 L11 P01 R10.0 ;**	Offset change

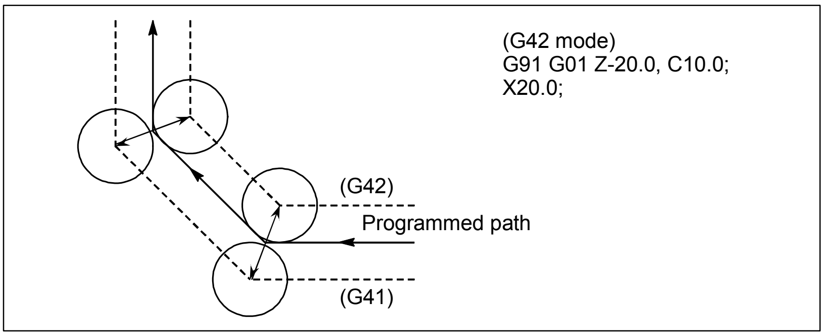
If the number of such blocks consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)), the tool arrives at the position vertical to this block at the end point of the previous block.
If the feed distance is 0 (<5>), this applies even if only one block is specified.



Overcutting may, therefore, occur in the above figure.

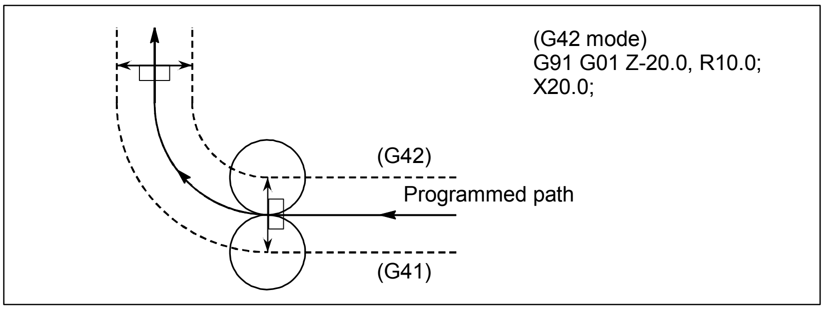
- Tool nose radius compensation when chamfering is performed

Movement after compensation is shown below.



- Tool nose radius compensation when a corner R is performed

Movement after compensation is shown below.



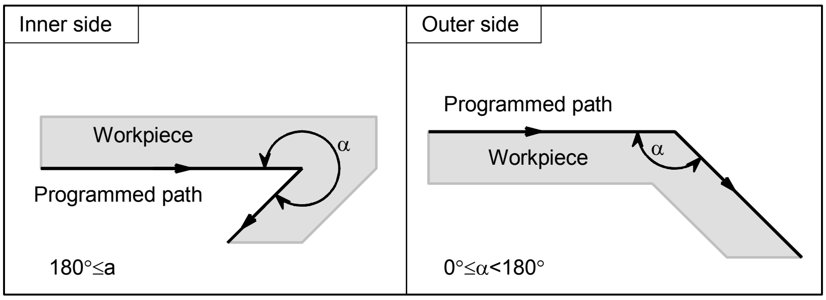
6.6 DETAILS OF CUTTER OR TOOL NOSE RADIUS COMPENSATION

6.6.1 Overview

The following explanation focuses on the cutter compensation, but applies to the tool nose radius compensation as well.

- Inner side and outer side

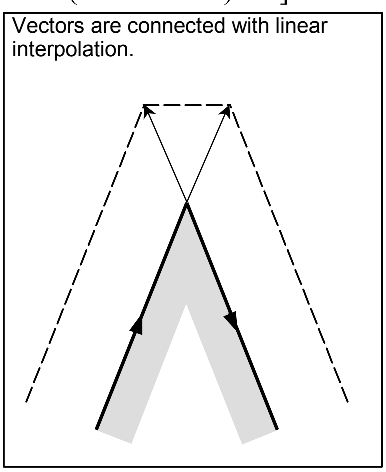
When an angle of intersection of the tool paths specified with move commands for two blocks on the workpiece side is over 180°, it is referred to as "inner side." When the angle is between 0° and 180°, it is referred to as "outer side."



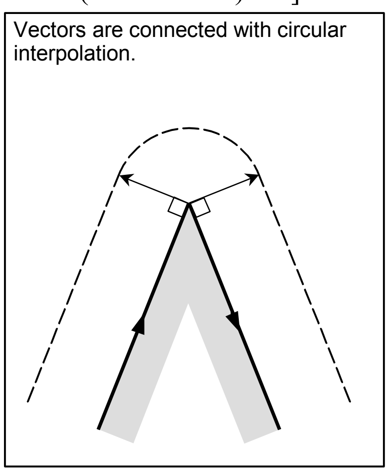
- Outer corner connection method

If the tool moves around an outer corner in cutter compensation mode, it is possible to specify whether to connect compensation vectors with linear interpolation or with circular interpolation, using parameter CCC (No. 19607#2).

<1> Linear connection type [Parameter CCC (No.19607#2) = 0]



<2> Circular connection type [Parameter CCC (No.19607#2) = 1]



- Cancel mode

The cutter compensation enters the cancel mode under the following conditions. (The system may not enter the cancel mode depending on the machine tool.)

<1> Immediately after the power is turned on
<2> When the <RESET> button on the MDI panel is pushed
<3> After a program is forced to end by executing M02 or M30
<4> After the cutter compensation cancel command (G40) is exercised

In the cancel mode, the compensation vector is set to zero, and the path of the center of tool coincides with the programmed path. A program must end in cancel mode. If it ends in the cutter compensation mode, the tool cannot be positioned at the end point, and the tool stops at a location the compensation vector length away from the end point.

- Start-up

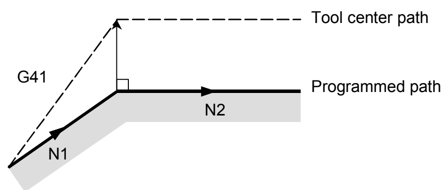
When a block which satisfies all the following conditions is executed in cancel mode, the CNC enters the cutter compensation mode. Control during this operation is called start-up.

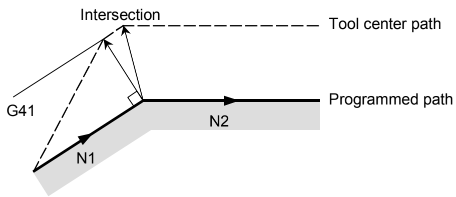
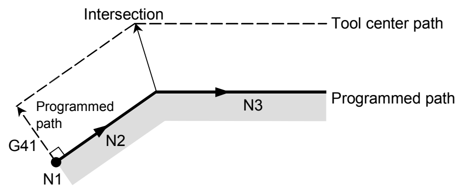
<1> G41 or G42 is contained in the block, or has been specified to place the CNC in the cutter compensation mode.
<2> 0 < compensation number of cutter compensation ≤ maximum compensation number
<3> Positioning (G00) or linear interpolation (G01) mode
<4> A compensation plane axis command with a travel distance of 0 (except start-up type C) is specified.

If start-up is specified in circular interpolation (G02, G03) mode, PS0034 will occur.

As a start-up operation, one of the three types A, B, and C can be selected by setting parameter SUP (No. 5003#0) and parameter SUV (No. 5003#1) appropriately. The operation to be performed if the tool moves around an inner side is of single type only.

Table 6.6.1 (a) Start-up/cancel operation

SUV	SUP	Type	Operation
0	0	Type A	A compensation vector is output, which is vertical to the block subsequent to the start-up block and the block preceding the cancel block. 

SUV	SUP	Type	Operation
0	1	Type B	A compensation vector is output, which is vertical to the start-up block and the cancel block. An intersection vector is also output. 
1	0 1	Type C	When the start-up block and the cancel block are blocks without tool movement, the tool moves by the cutter or tool nose radius compensation value in the direction vertical to the block subsequent to the start-up block and the block preceding the cancel block.  For a block with tool movement, the tool follows the SUP setting: If it is 0, type A is assumed and if 1, type B is assumed.

- Reading input commands in cutter compensation mode

In cutter compensation mode, input commands are read from usually three blocks and up to eight blocks depending on the setting of parameter (No. 19625) to perform intersection calculation or an interference check, described later, regardless of whether the blocks are with or without tool movement, until a cancel command is received.
To perform intersection calculation, it is necessary to read at least two blocks with tool movement. To perform an interference check, it is necessary to read at least three blocks with tool movement.

As the setting of parameter (No. 19625), that is, the number of blocks to read, increases, it is possible to predict overcutting (interference) for up to more subsequent commands. Increases in blocks to read and analyze, however, cause reading and analysis to take more time.

- Ending (canceling) cutter compensation

In cutter compensation mode, cutter compensation is canceled if a block that satisfies at least either one of the following conditions is executed:

<1> G40 is specified.

<2> D00 is specified as the compensation number of cutter compensation.

If cutter compensation cancel is to be performed, it must not be by a circular command (G02, G03). Otherwise, an alarm will occur.

For a cancel operation, one of three types, A, B, and C, can be selected by appropriately setting parameter SUP (No. 5003#0) and parameter SUV (No. 5003#1). The operation to be performed if the tool turns around the inside is of a single type.

- Meaning of symbols

The following symbols are used in subsequent figures:

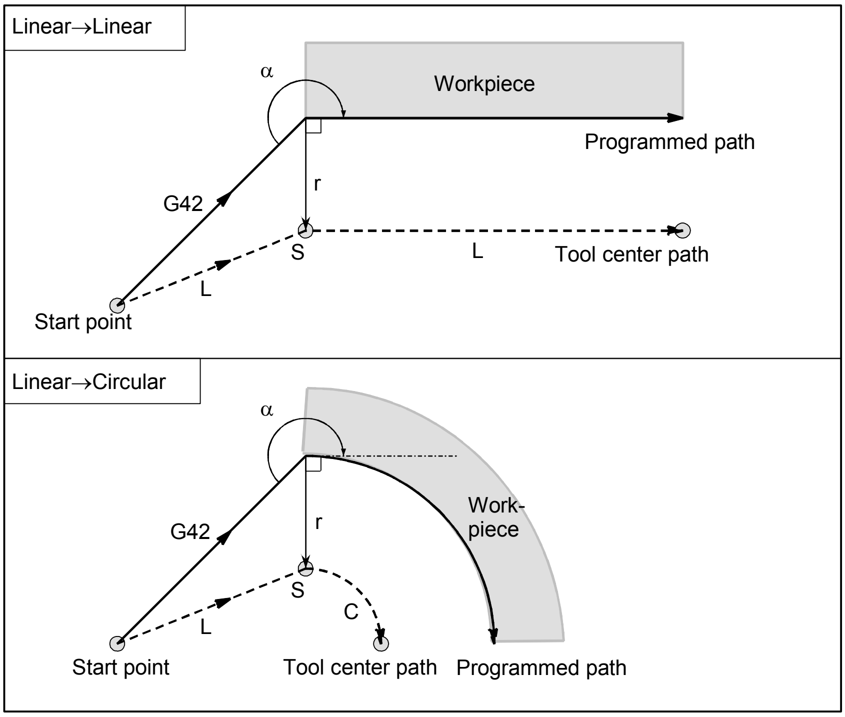
- S indicates a position at which a single block is executed once.
- SS indicates a position at which a single block is executed twice.
- SSS indicates a position at which a single block is executed three times.
- L indicates that the tool moves along a straight line.
- C indicates that the tool moves along an arc.
- r indicates the cutter or tool nose radius compensation value.
- An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by r.
- ○ indicates the center of the tool.

6.6.2 Tool Movement in Start-up

When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start-up):

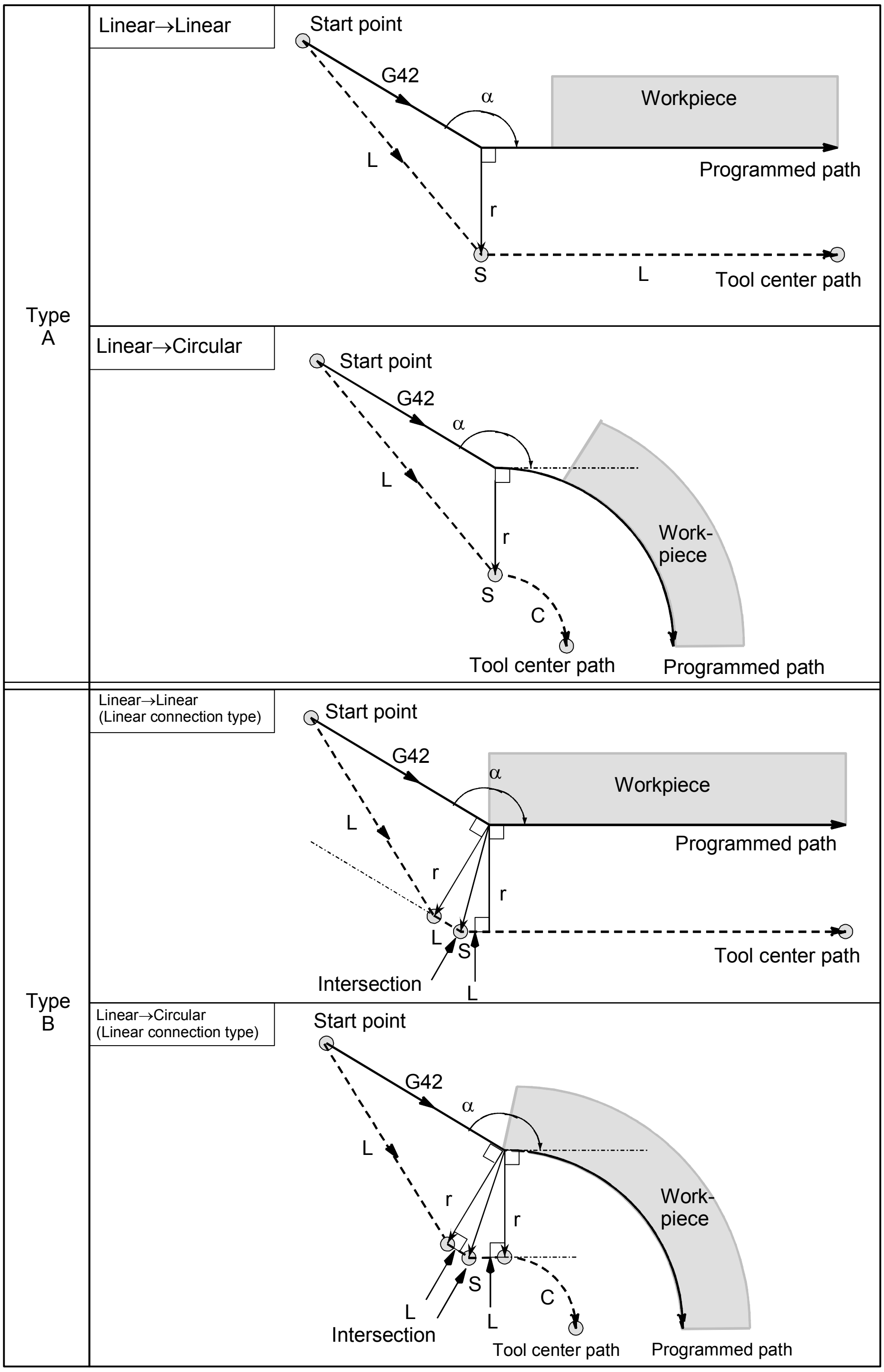
Explanation

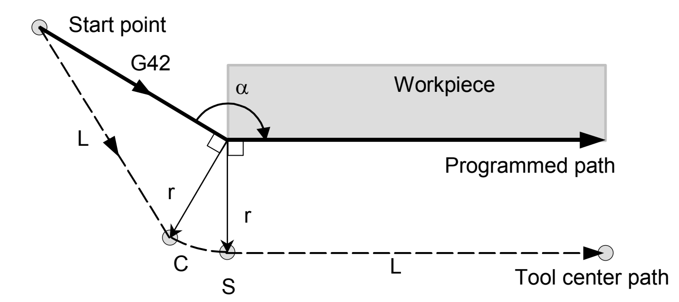
- Tool movement around an inner side of a corner (180°≤ α)

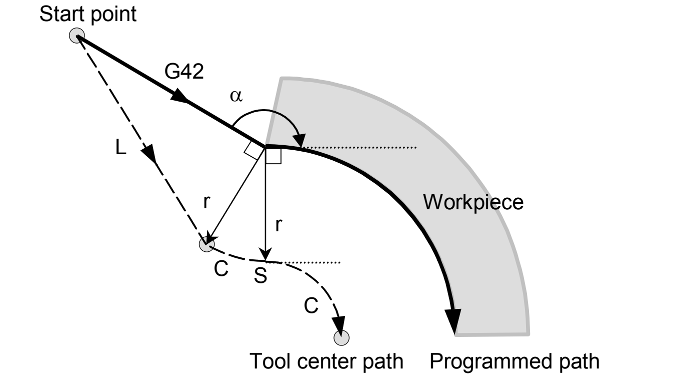


- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an obtuse angle (90°≤ α<180°)

Tool path in start-up has two types A and B, and they are selected by parameter SUP (No.5003#0).



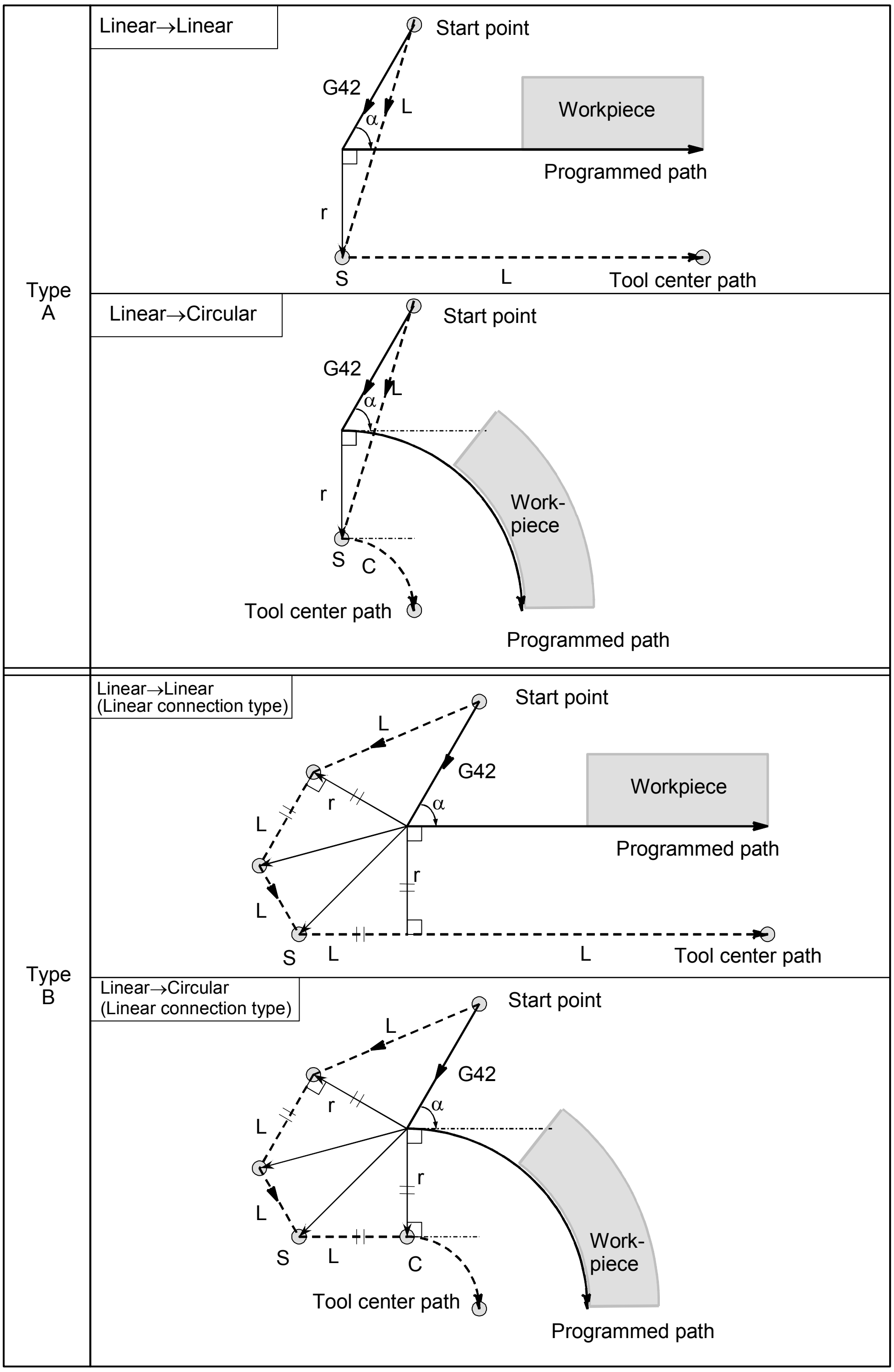
<table>
<tr>
<td rowspan="2">Type B</td>
<td>Linear→Linear (Circular connection type)</td>
<td>


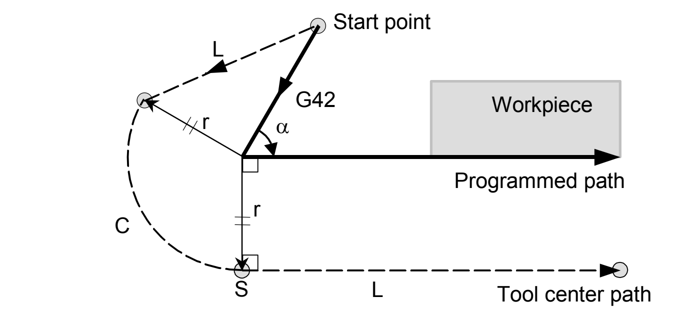
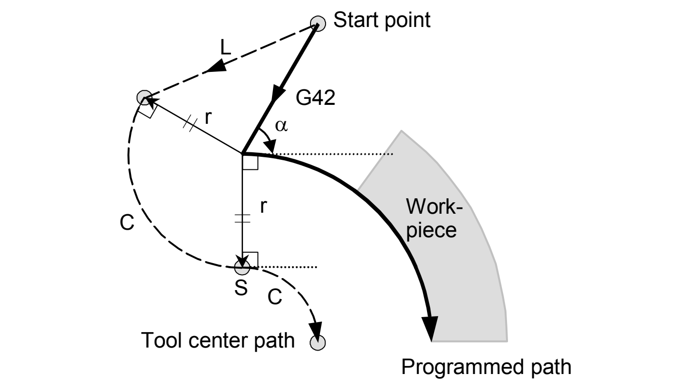
</td>
</tr>
<tr>
<td>Linear→Circular (Circular connection type)</td>
<td>


</td>
</tr>
</table>

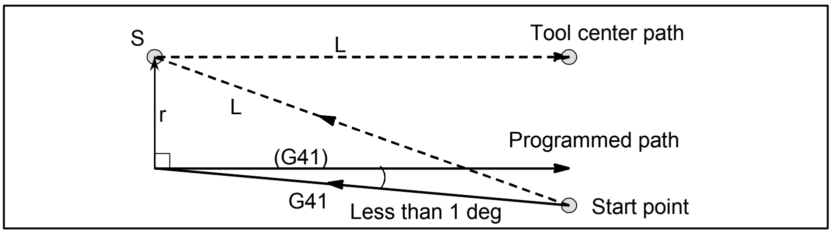
- Cases in which the start-up block is a block with tool movement and the tool moves around the outside at an acute angle (α<90°)

Tool path in start-up has two types A and B, and they are selected by parameter SUP (No.5003#0).



Type B	Linear→Linear (Circular connection type)	
	Linear→Circular (Circular connection type)	

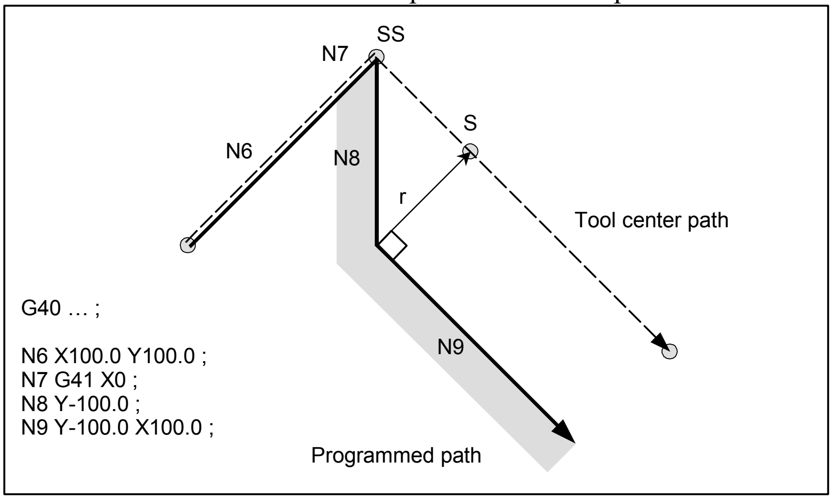
- **Tool movement around the outside linear → linear at an acute angle less than 1 degree (α<1°)**



- A block without tool movement specified at start-up

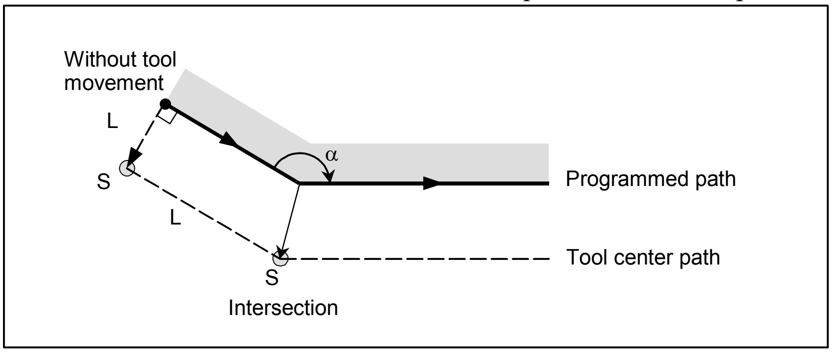
For type A and B

If the command is specified at start-up, the offset vector is not created. The tool does not operate in a start-up block.



For type C

The tool shifts by the compensation value in the direction vertical to the block with tool movement subsequent to the start-up block.



6.6.3 Tool Movement in Offset Mode

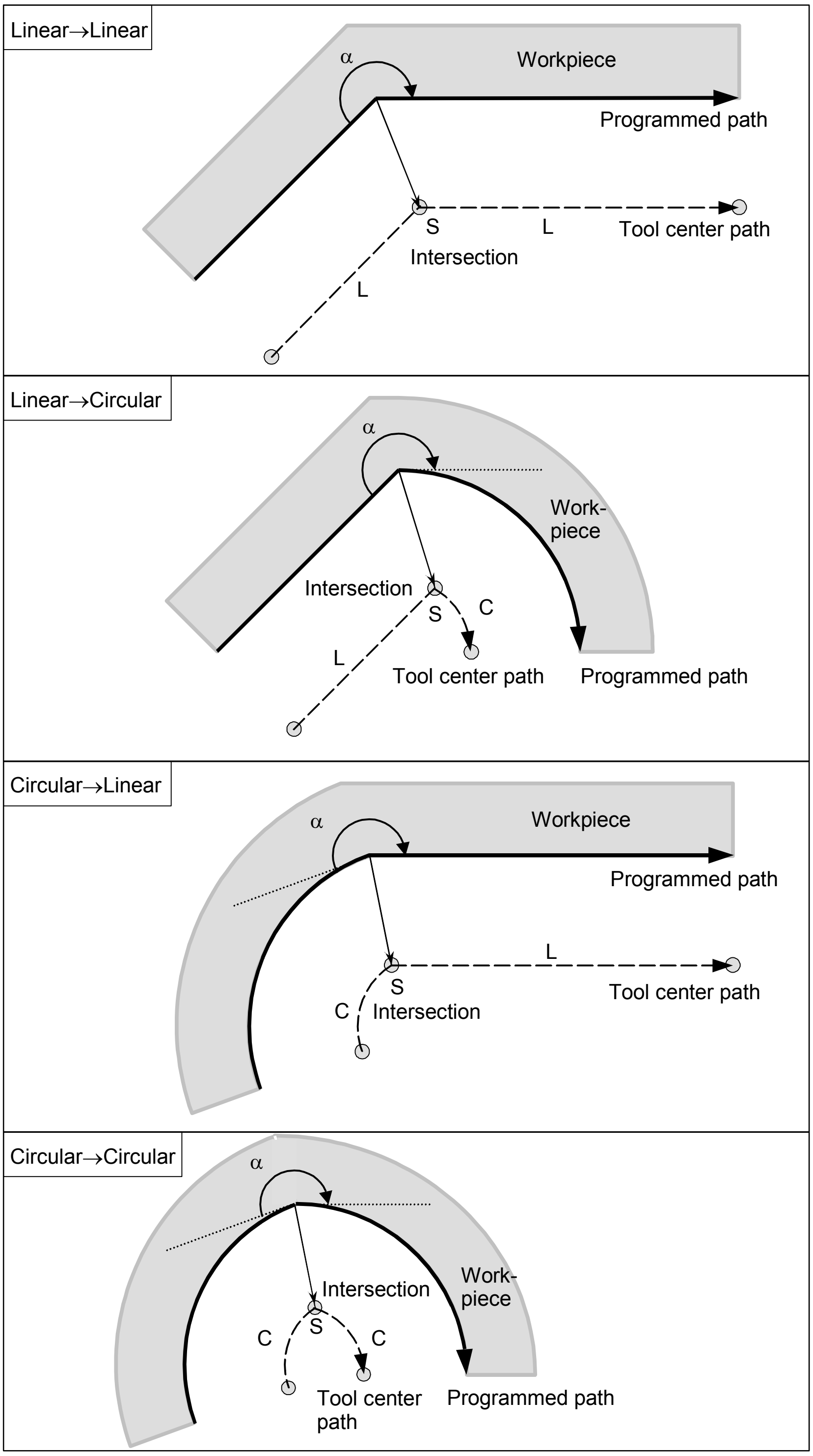
In offset mode, compensation is performed even for positioning commands, not to speak of linear and circular interpolations. To perform intersection calculation, it is necessary to read at least two blocks with tool movement. If, therefore, two or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as auxiliary function independent commands and dwell, are specified in succession, excessive or insufficient cutting may occur because intersection calculation fails. Assuming the number of blocks to read in offset mode, which is determined by parameter (No. 19625), to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which intersection calculation is possible is $(N - 2) \geq M$. For example, if the maximum number of blocks to read in offset mode is 5, intersection calculation is possible even if up to three blocks without tool movement are specified.

NOTE

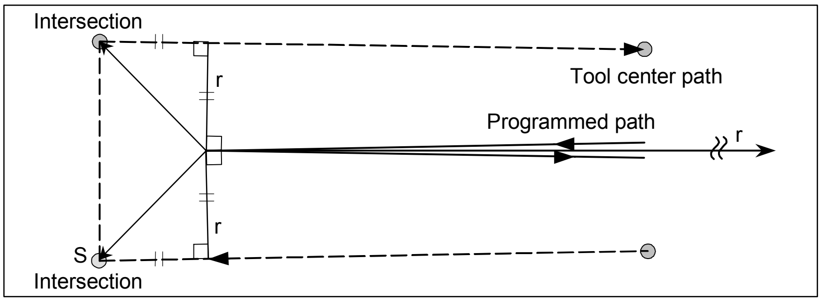
The condition necessary for an interference check, described later, differs from this condition. For details, see the explanation of the interference check.

If a G or M code in which buffering is suppressed is specified, no subsequent commands can be read before that block is executed, regardless of the setting of parameter (No. 19625). Excessive or insufficient cutting may, therefore, occur because of an intersection calculation failure.

- Tool movement around the inside of a corner (180°≤ α)

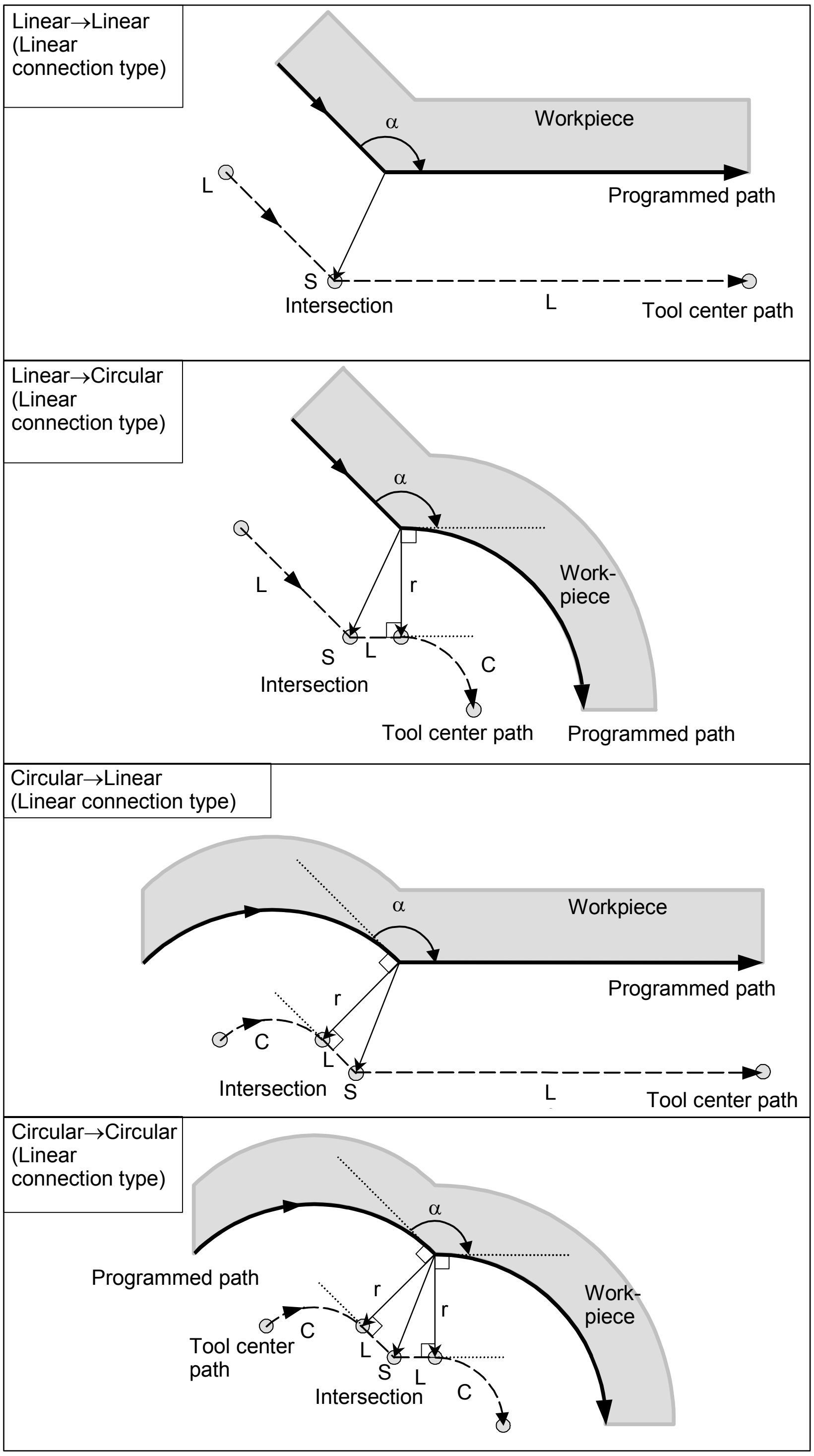


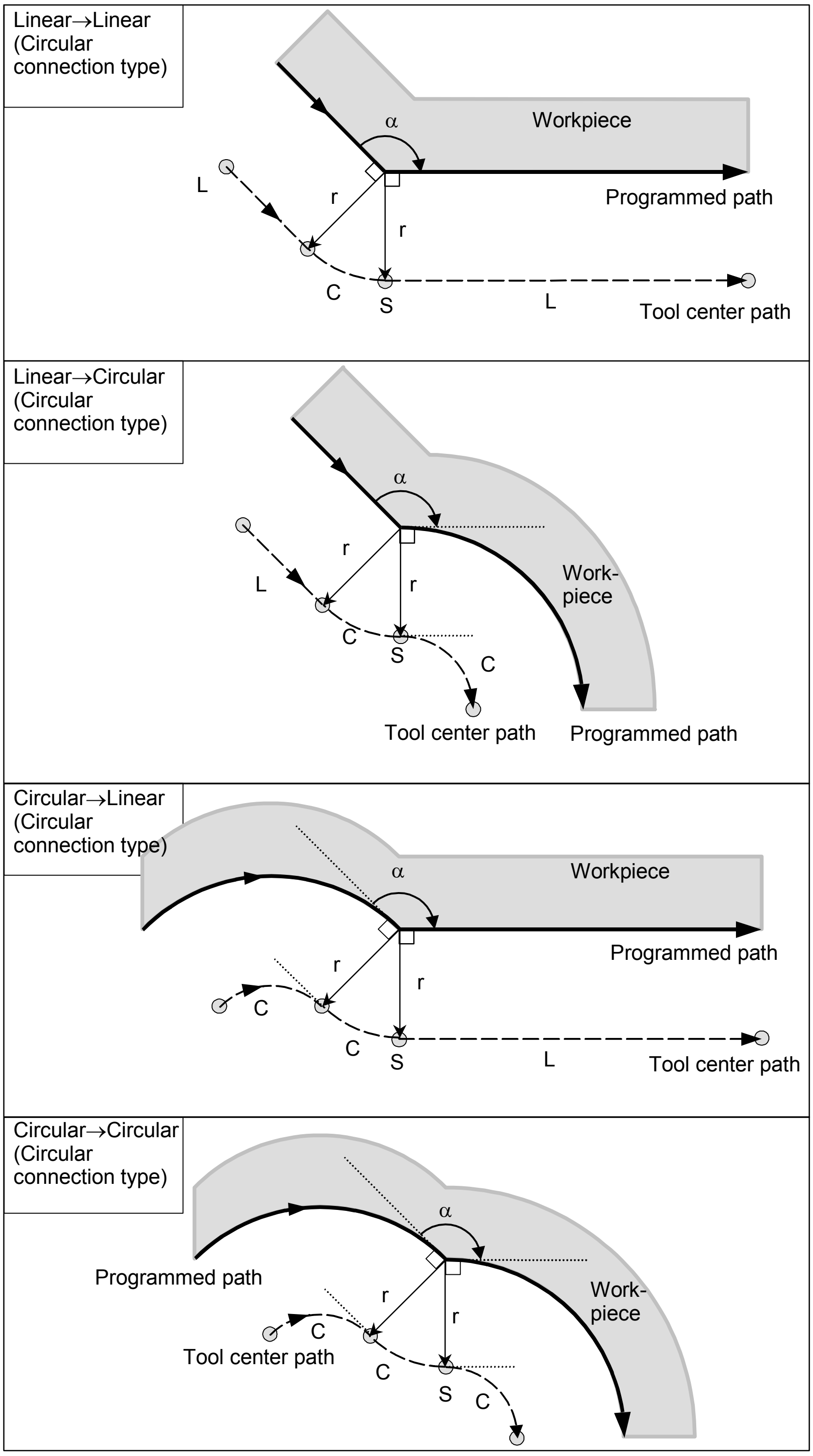
- Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear



Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

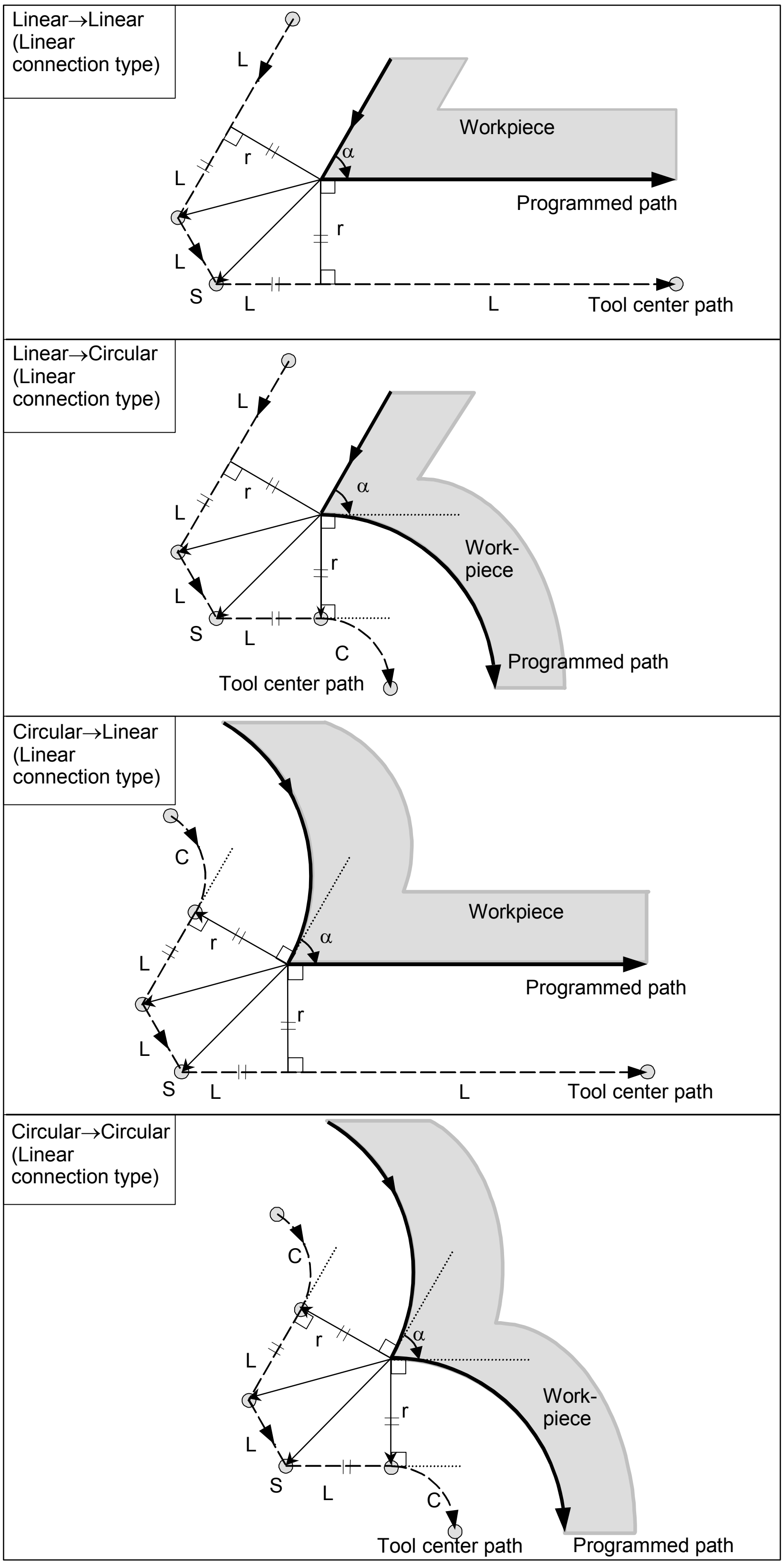
- Tool movement around the outside corner at an obtuse angle (90°≤α<180°)

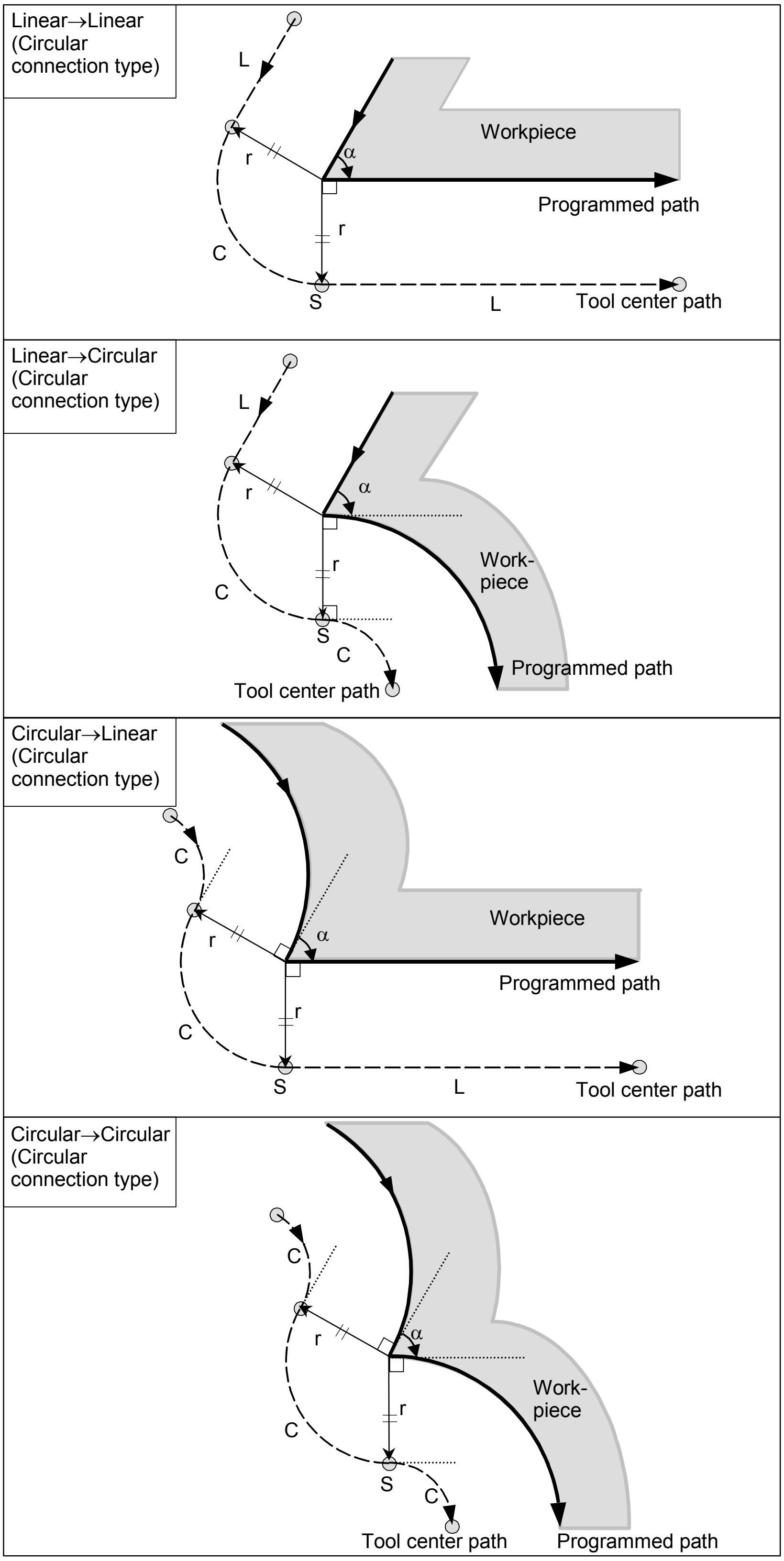




Linear→Linear
(Circular
connection type)
Workpiece
α
L
r
r
Programmed path
C
S
L
Tool center path
Linear→Circular
(Circular
connection type)
α
r
L
r
Work-
piece
C
S
C
Tool center path
Programmed path
Circular→Linear
(Circular
connection type)
α
Workpiece
Programmed path
r
r
C
C
S
L
Tool center path
Circular→Circular
(Circular
connection type)
α
Programmed path
r
r
Work-
piece
C
Tool center path
C
S
C

- Tool movement around the outside corner at an acute angle (α<90°)

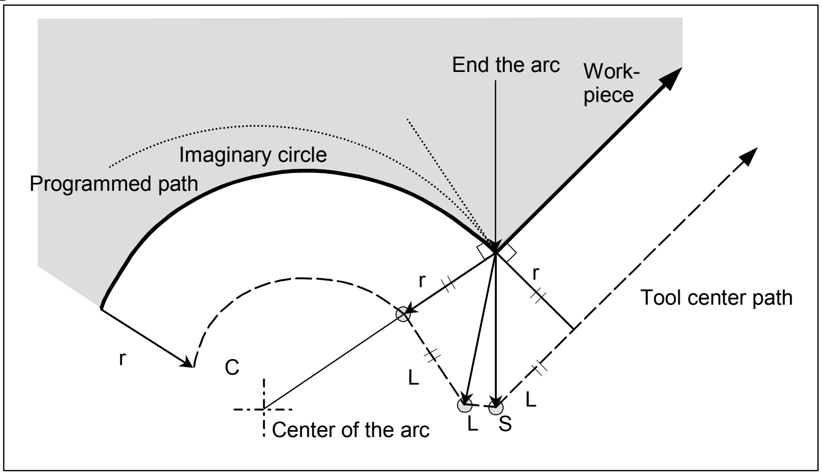



Linear→Linear (Circular connection type)
L
Workpiece
α
r
Programmed path
r
C
S
L
Tool center path
Linear→Circular (Circular connection type)
L
α
r
r
Work-piece
C
S
C
Programmed path
Tool center path
Circular→Linear (Circular connection type)
C
Workpiece
r
α
Programmed path
r
C
S
L
Tool center path
Circular→Circular (Circular connection type)
C
r
α
Work-piece
r
C
S
C
Tool center path
Programmed path

- When it is exceptional

End point for the arc is not on the arc

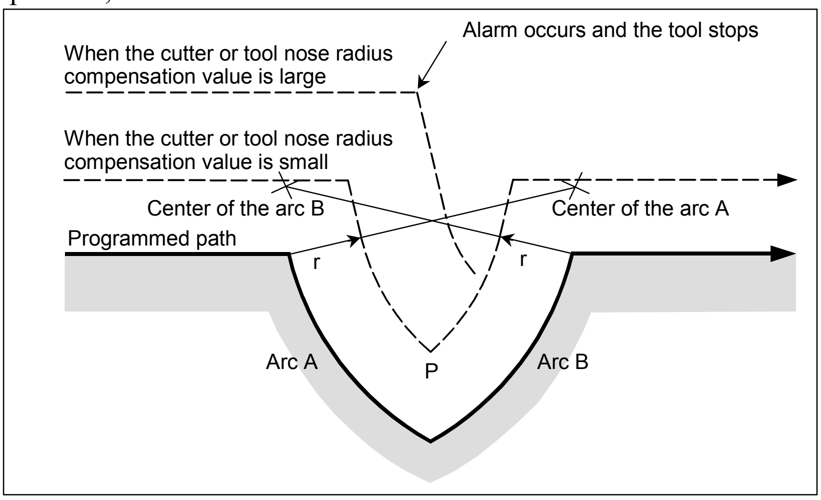
If the end of a line leading to an arc is not on the arc as illustrated below, the system assumes that the cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end point. Based on this assumption, the system creates a vector and carries out compensation. The same description applies to tool movement between two circular paths.



There is no inner intersection

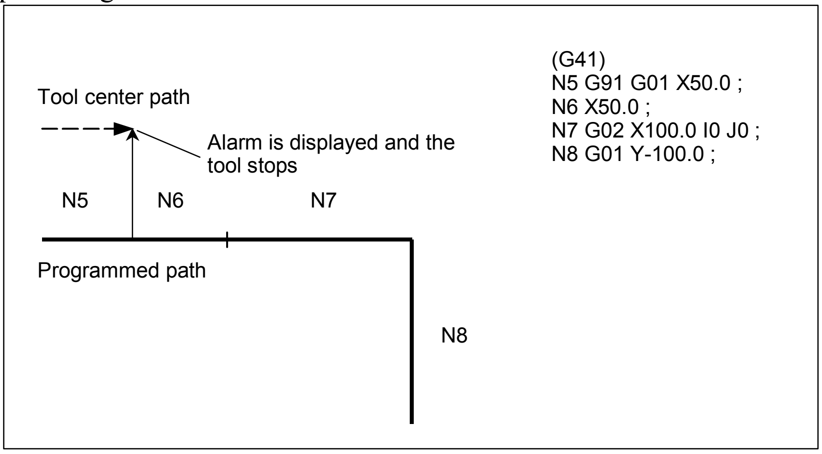
If the cutter or tool nose radius compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for cutter or tool nose radius compensation. When this is predicted, PS0033 occurs at the end of the previous block and the tool is stopped.

In the Example shown below, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for cutter or tool nose radius compensation. If an excessively large value is specified, this intersection does not occur.



- When the center of the arc is identical with the start point or the end point

If the center of the arc is identical with the start point or end point, PS0041 is displayed, and the tool will stop at the start point of the preceding block of the arc.



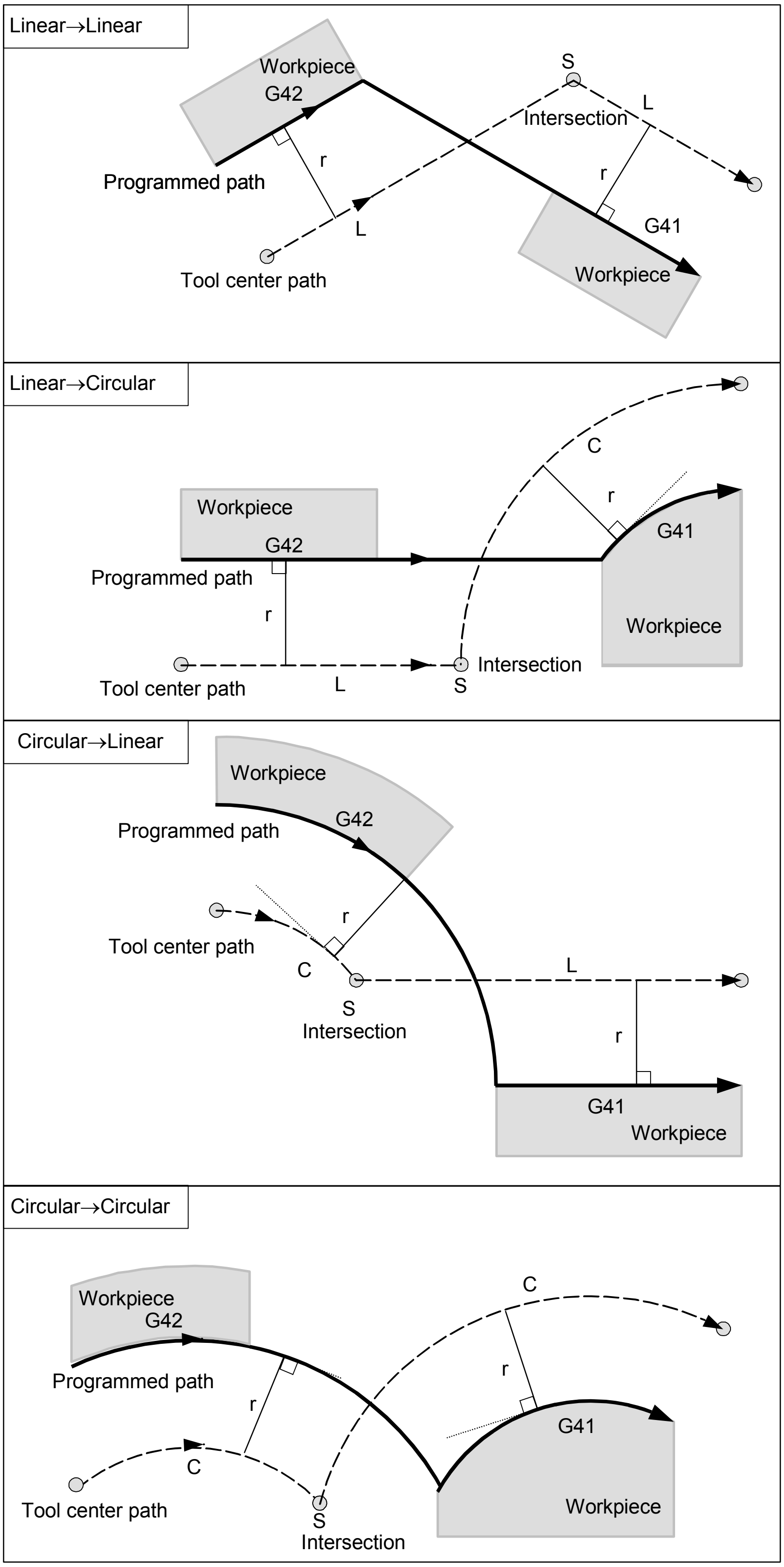
- Change in the offset direction in the offset mode

The offset direction is decided by G codes (G41 and G42) for cutter or tool nose radius compensation and the sign of the compensation value as follows.

G code \ Sign of compensation	+	–
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

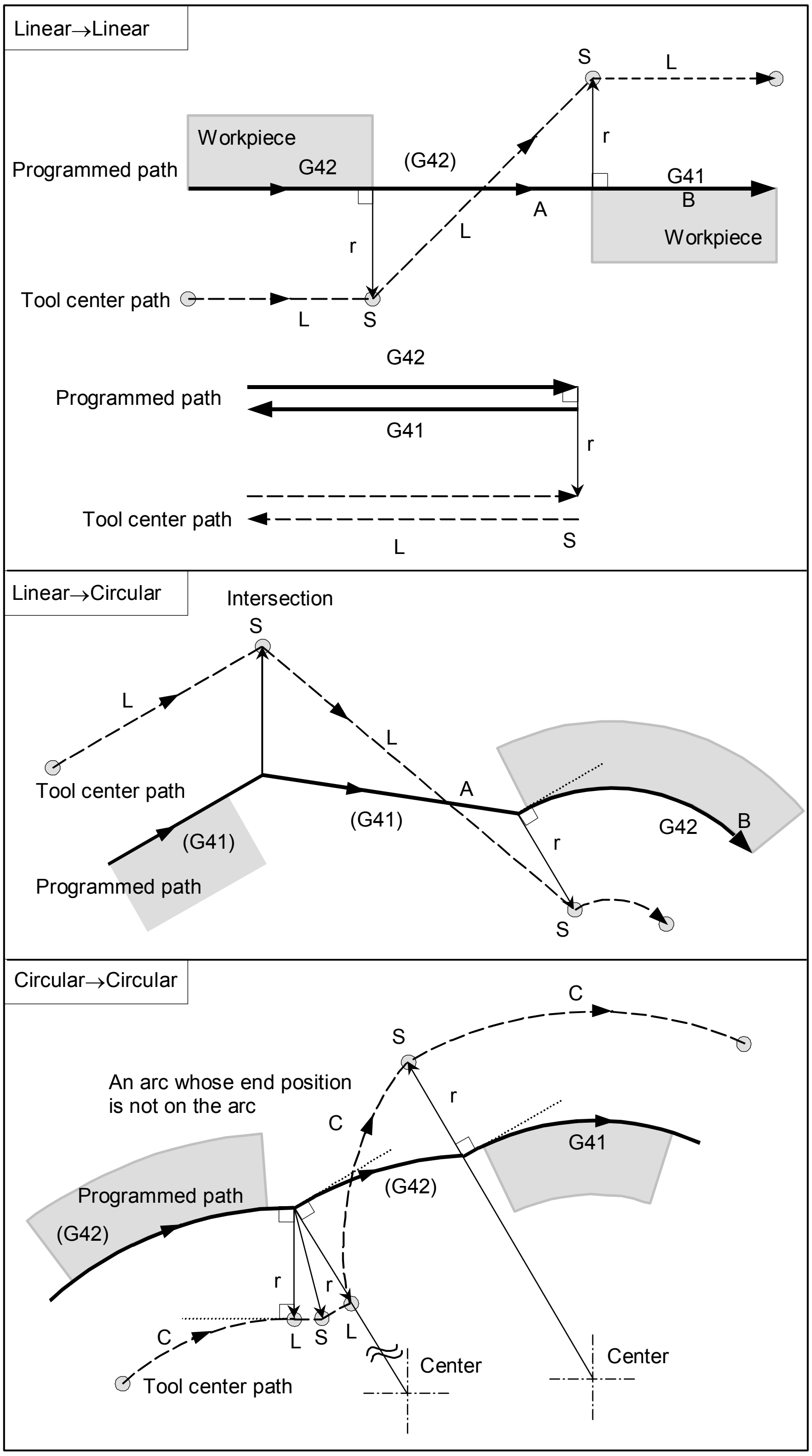
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block.
However, the change is not available in the start-up block and the block following it.

- Tool center path with an intersection



- Tool center path without an intersection

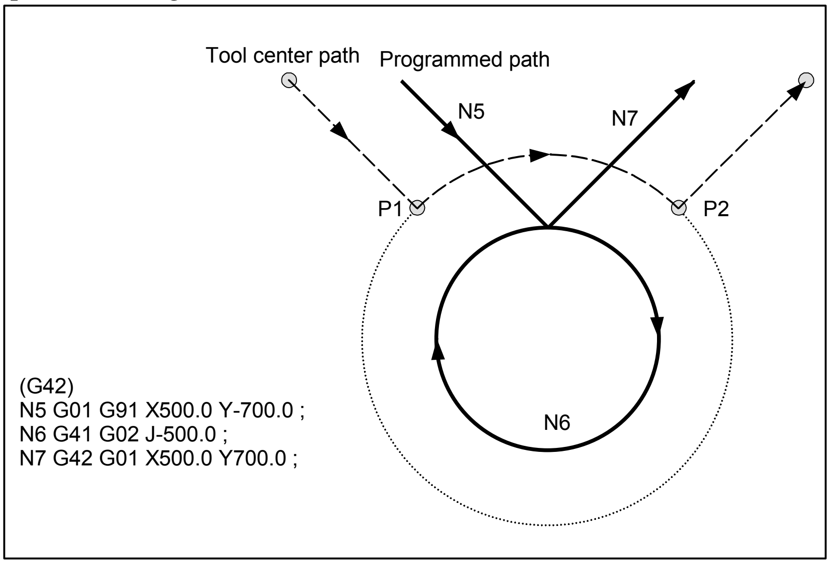
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



- The length of tool center path larger than the circumference of a circle

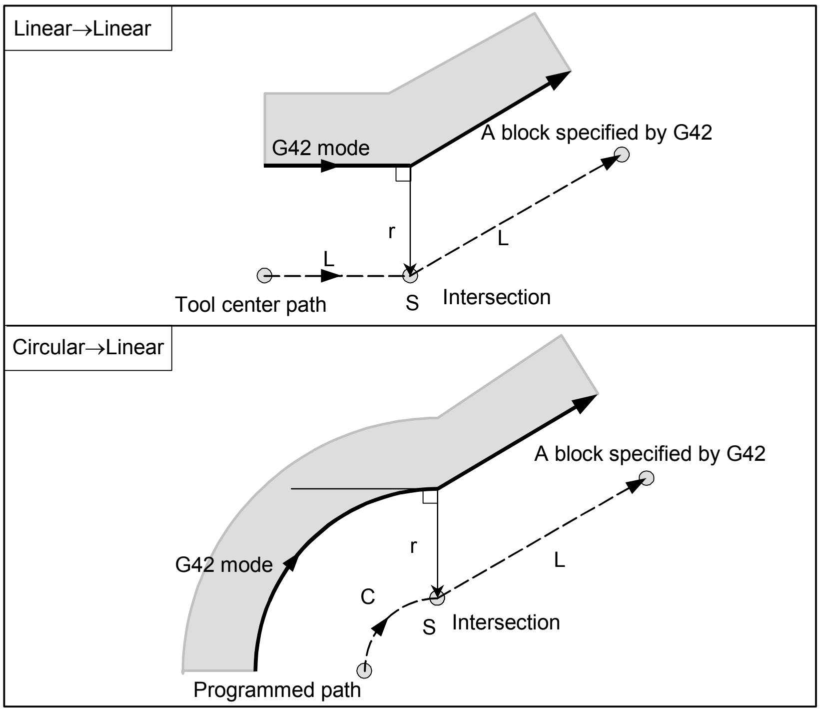
Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.

In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P_1 to P_2 as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.



- Cutter compensation G code in the offset mode

The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the cutter compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.
When the direction of offset is expected to be changed by the command of cutter compensation G code (G41, G42), see "Change in the offset direction in the offset mode".

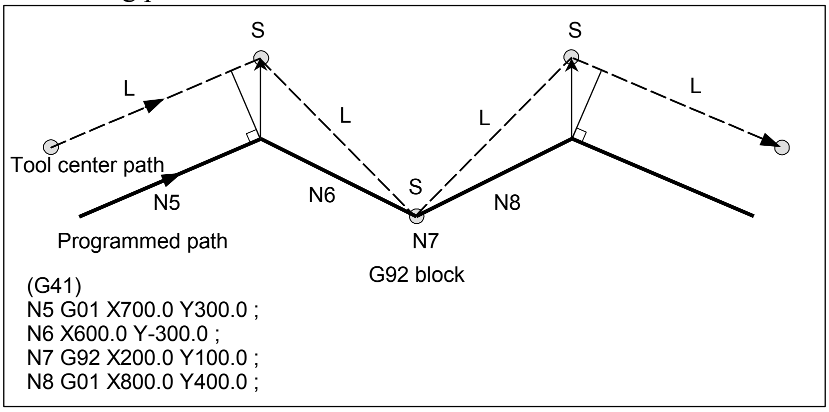


- Command canceling the offset vector temporarily

During offset mode, if G92 (workpiece coordinate system setting) or G52 (local coordinate system setting) is commanded, the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.

In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled.

Also when restored to offset mode, the tool moves directly to the intersecting point.



Before specifying G28 (reference position return), G29 (movement from reference position), G30 (second, third, and fourth reference position return), G30.1 (floating reference position return), and G53 (machine coordinate system selection) commands, cancel offset mode, using G40. If an attempt is made to specify any of the commands in offset mode, the offset vector temporarily disappears.

- If I, J, and K are specified in a G00/G01 mode block

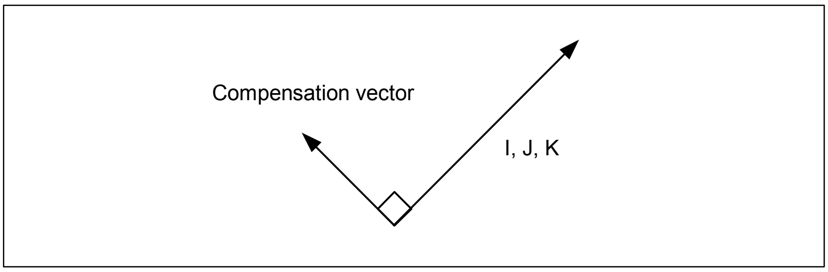
At the start of cutter compensation or in that mode, by specifying I, J, and K in a positioning mode (G00) or linear interpolation mode (G01) block, it is possible to set the compensation vector at the end point of that block in the direction vertical to that specified by I, J, and K. This makes it possible to change the compensation direction intentionally.

IJ type vector (XY plane)

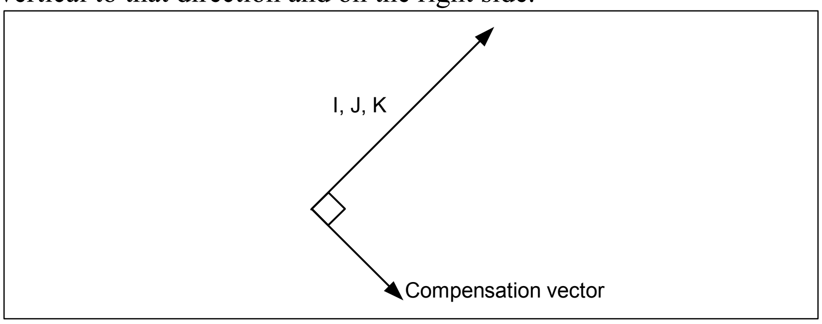
The following explains the compensation vector (IJ type vector) to be created on the XY compensation plane (G17 mode). (The same explanation applies to the KI type vector on the G18 plane and the JK type vector on the G19 plane.) As shown in the figure below, it is assumed that the compensation vector (IJ type vector) is the vector with a size equal to the compensation value and vertical to the direction specified by I and J, without performing intersection calculation on the programmed path. I and J can be specified both at the start of cutter compensation and in that mode. If they are specified at the start of compensation, any start-up type set in the appropriate parameter will be invalid, and an IJ type vector is assumed.

Offset vector direction

In G41 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the left side.



In G42 mode, the direction specified by I, J, and K is assumed an imaginary tool movement direction, and an offset vector is created vertical to that direction and on the right side.

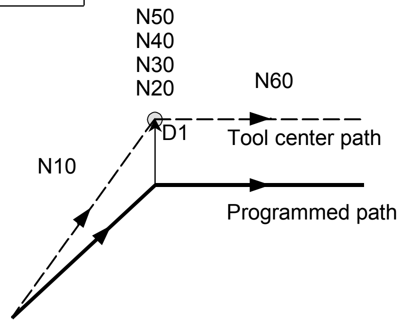


Example

If I and J are specified at the start of compensation (with tool movement)

```
    (G40)
N10 G91 G41 X100.0 Y100.0
      I1 D1 ;
N20 G04 X1000 ;
N30 G01 F1000 ;
N40 S300 ;
N50 M50 ;
N60 X150. ;
```

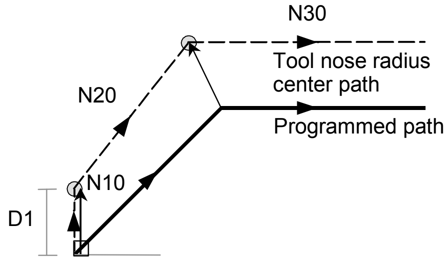
Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.



If I and J are specified at the start of compensation (without tool movement)

```
    (G40)
N10 G41 I1 D1 ;
N20 G91 X100. Y100. ;
N30 X150. ;
```

Note) In N10, a vector is specified with a size of D1 in the direction vertical to the X axis, using I1.

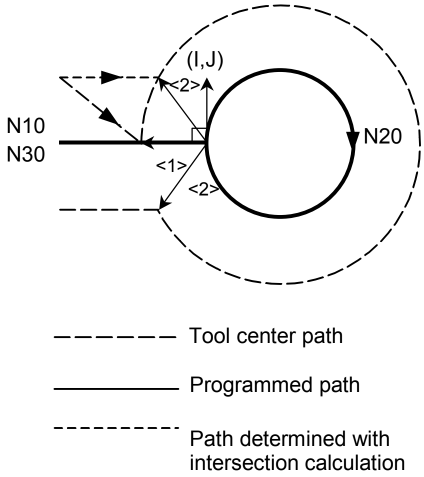


If I and J are specified at the start of compensation (with tool movement)

```
    (G17 G41 G91 D1)
N10 G00 X150. J50. ;
N20 G02 I50. ;
N30 G00 X-150. ;
```

Note) In N10, a vector is specified with a size of D1 in the direction vertical to the Y axis, using J50.

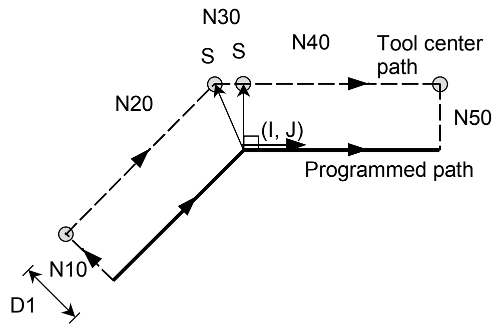
<1> IJ type vector
<2> Vector determined with intersection calculation



If I and J are specified in a block without tool movement in compensation mode

Start-up/cancel type C

```
N10 G41 D1 G01 F1000 ;
N20 G91 X100. Y100. ;
N30 I10. ;
N40 X150. ;
N50 G40 ;
```

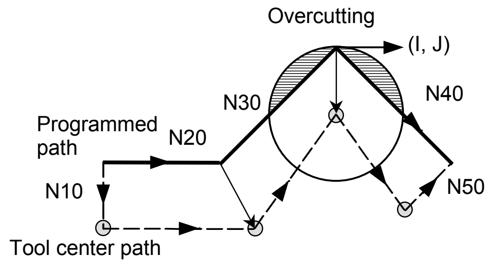


Limitation

If an IJ type vector is specified, tool interference may occur due to that vector alone, depending on the direction. If this occurs, no interference alarm will occur, or no interference avoidance will be performed. Overcutting may, therefore, result.

Start-up/cancel
Type C

```
N10 G42 D1 F1000 ;
N20 G91 X100. ;
N30 X100. Y-100. I10. ;
N40 X100. Y-100. ;
N50 G40 ;
```



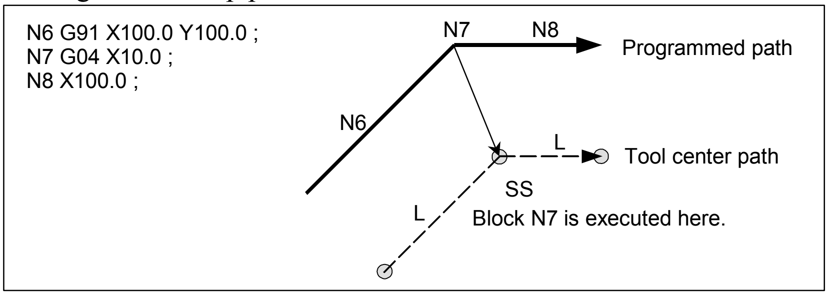
- A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

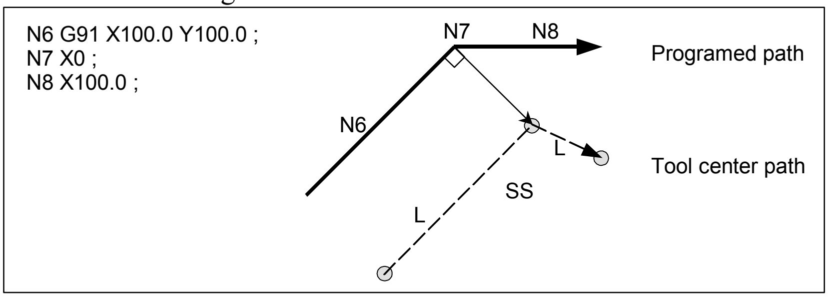
M05 ;	:	M code output
S21 ;	:	S code output
G04 X10.0 ;	:	Dwell
G22 X100000 ;	:	Machining area setting
G10 L11 P01 R10.0 ;	:	Cutter compensation value setting/changing
(G17) Z200.0 ;	:	Move command not included in the offset plane.
G90 ;, O10 ;, N20 ;	:	G, O, and N codes only
G91 X0 ;	:	Move distance is zero.

- A block without tool movement specified in offset mode

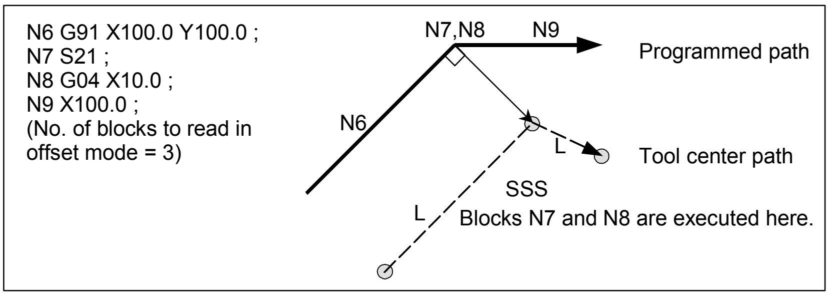
Unless the number of blocks without movement consecutively specified is more than N-2 blocks (where N is the number of blocks to read in offset mode (parameter No. 19625)) in offset mode, the vector and the tool center path will be as usual. This block is executed at the single block stop point.



For an axis command for which the move distance is zero, however, a vector with a size equal to the compensation value will be created vertical to the movement direction in the previous block, even if the number of block is 1. Note that specifying such a command may result in overcutting.



In offset mode, the number of blocks without movement consecutively specified must not exceed N-2 (where N is the number of blocks to read in offset mode (parameter (No. 19625)). If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.

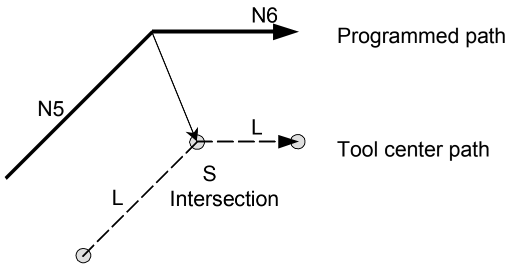


- If an M/G code that suppresses buffering is specified

If an M/G code that suppresses buffering is specified in offset mode, it is no longer possible to read and analyze subsequent blocks regardless of the number of blocks to read in offset mode, which is determined by parameter (No. 19625). Then, intersection calculation and a interference check, described later, are no longer possible. If this occurs, overcutting may occur because a vertical vector is output in the immediately preceding block.

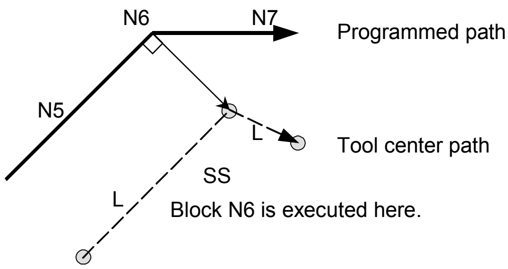
If an M code (M50) that suppresses buffering is not specified

```
  (G42)
N5 G91 G01 X40.0 Y40.0 ;
N6 X40.0 ;
     :
     :
```



If an M code (M50) that suppresses buffering is specified

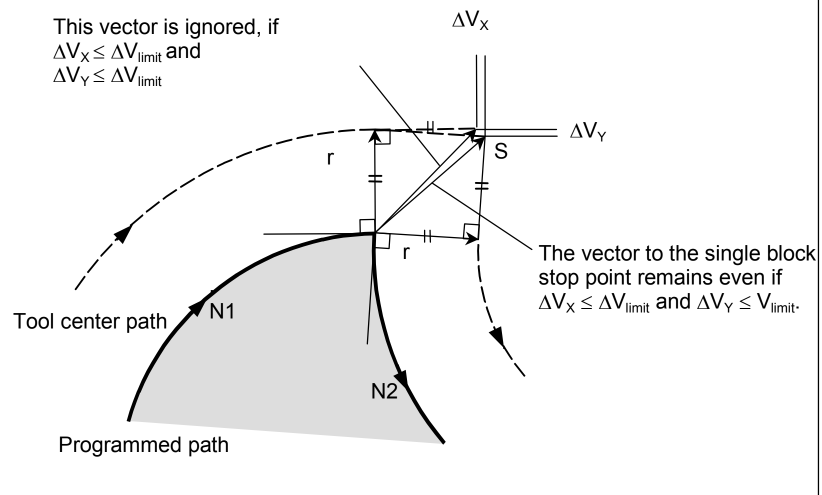
```
  (G42)
N5 G91 G01 X40.0 Y40.0 ;
N6 M50 ;
N7 X40.0 ;
     :
     :
```



- Corner movement

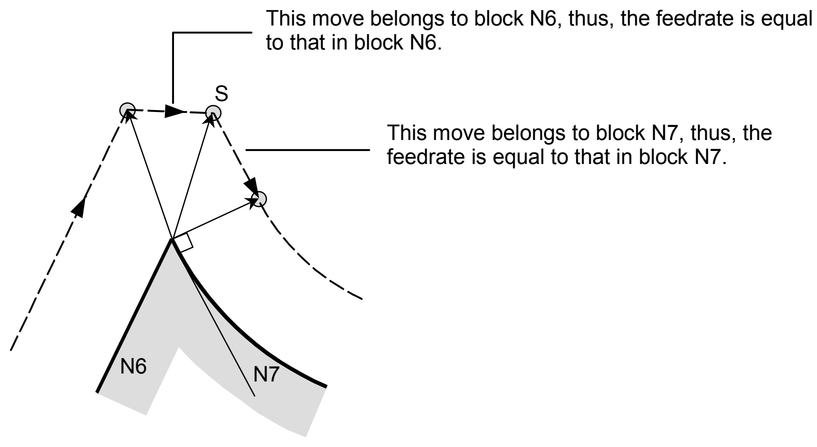
When two or more offset vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.

If these vectors almost coincide with each other (the distance of corner movement between the vectors is judged short due to the setting of parameter (No. 5010)), corner movement is not performed. In this case, the vector to the single block stop point takes precedence and remains, while other vectors are ignored. This makes it possible to ignore the very small movements arising from performing cutter compensation, thereby preventing velocity changes due to interruption of buffering.



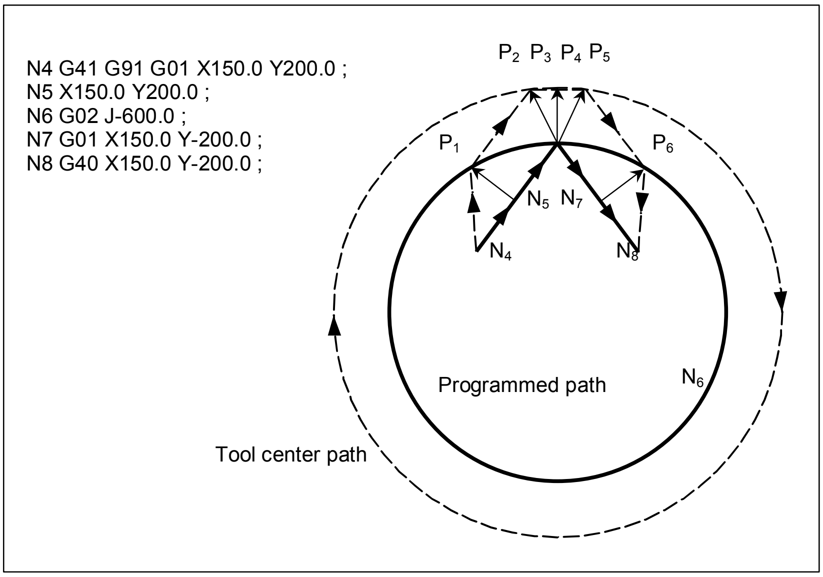
ΔV_{limit} is determined with the setting of parameter (No. 5010).

If the vectors are not judged to almost coincide (therefore, are not erased), movement to turn around the corner is performed. The corner movement that precedes the single block stop point belongs to the previous block, while the corner movement that succeeds the single block stop point belongs to the latter block.



However, if the path of the next block is semicircular or more, the above function is not performed.

The reason for this is as follows:



If the vector is not ignored, the tool path is as follows:
$P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow$ (Circle) $\rightarrow P_4 \rightarrow P_5 \rightarrow P_6$
But if the distance between P_2 and P_3 is negligible, the point P_3 is ignored. Therefore, the tool path is as follows:
$P_2 \rightarrow P_4$
Namely, circle cutting by the block N6 is ignored.

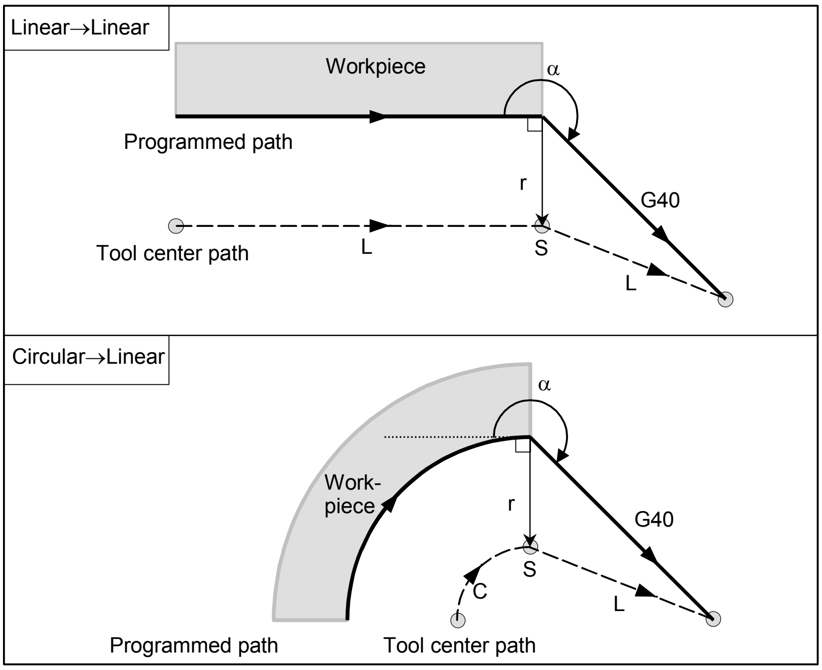
- Interruption of manual operation

For manual operation during the offset mode, see "Manual Absolute ON and OFF."

6.6.4 Tool Movement in Offset Mode Cancel

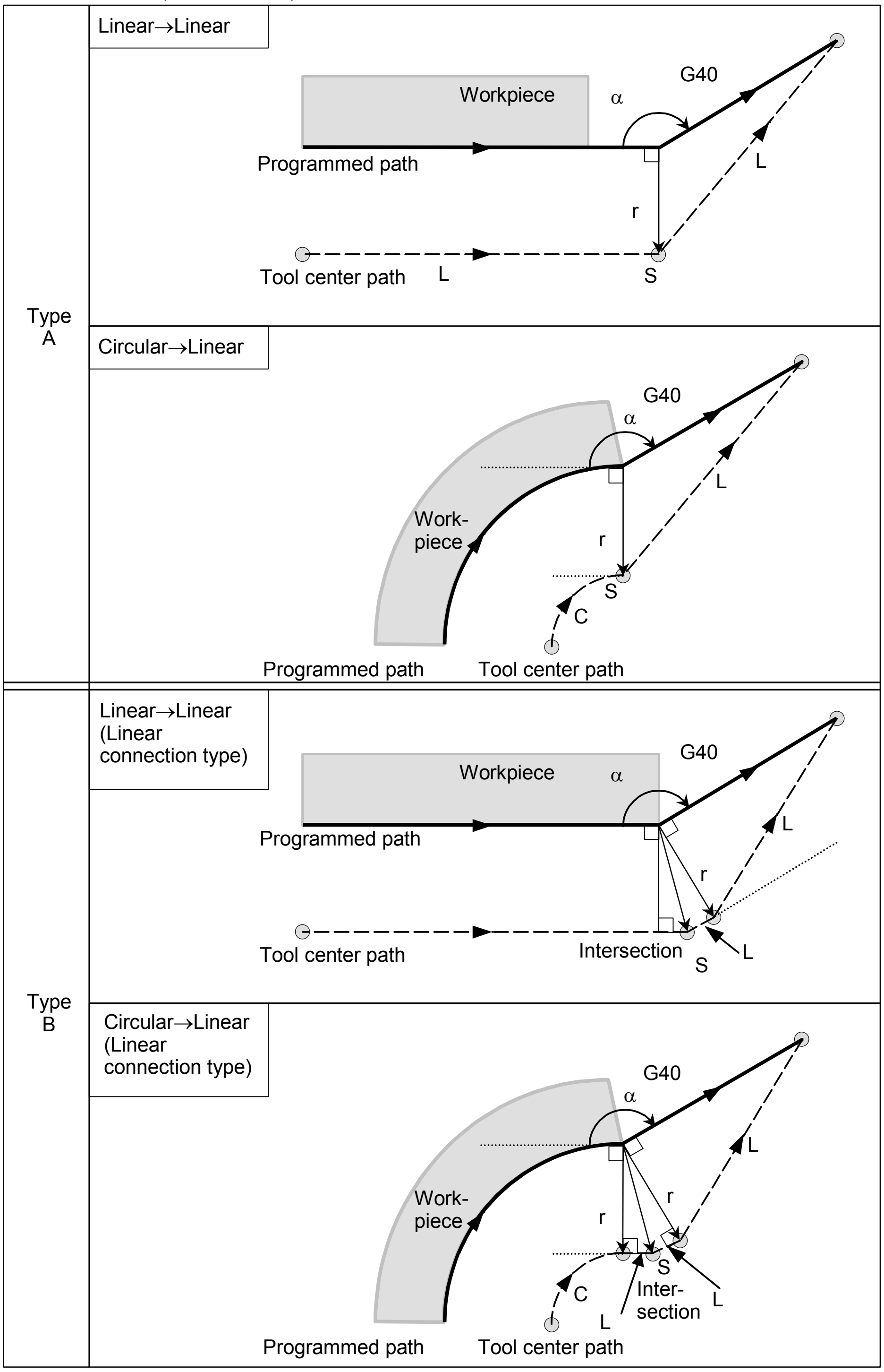
Explanation

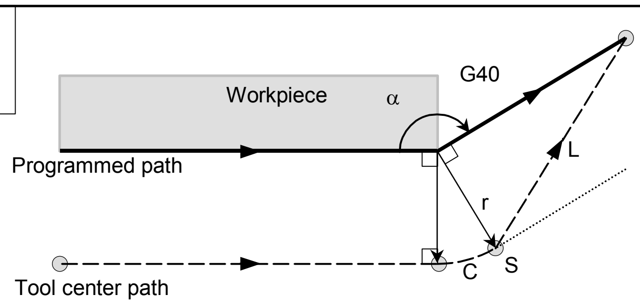
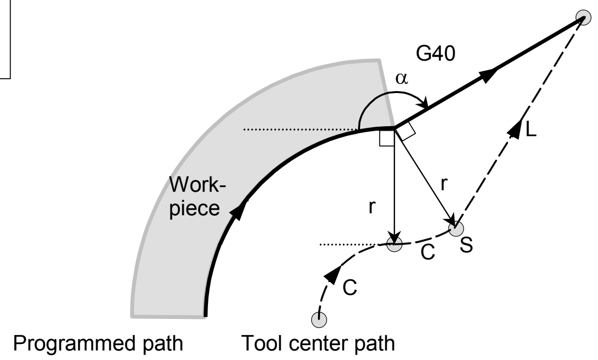
- If the cancel block is a block with tool movement, and the tool moves around the inside (180° ≤ α)



- If the cancel block is a block with tool movement, and the tool moves around the outside at an obtuse angle (90° ≤ α < 180°)

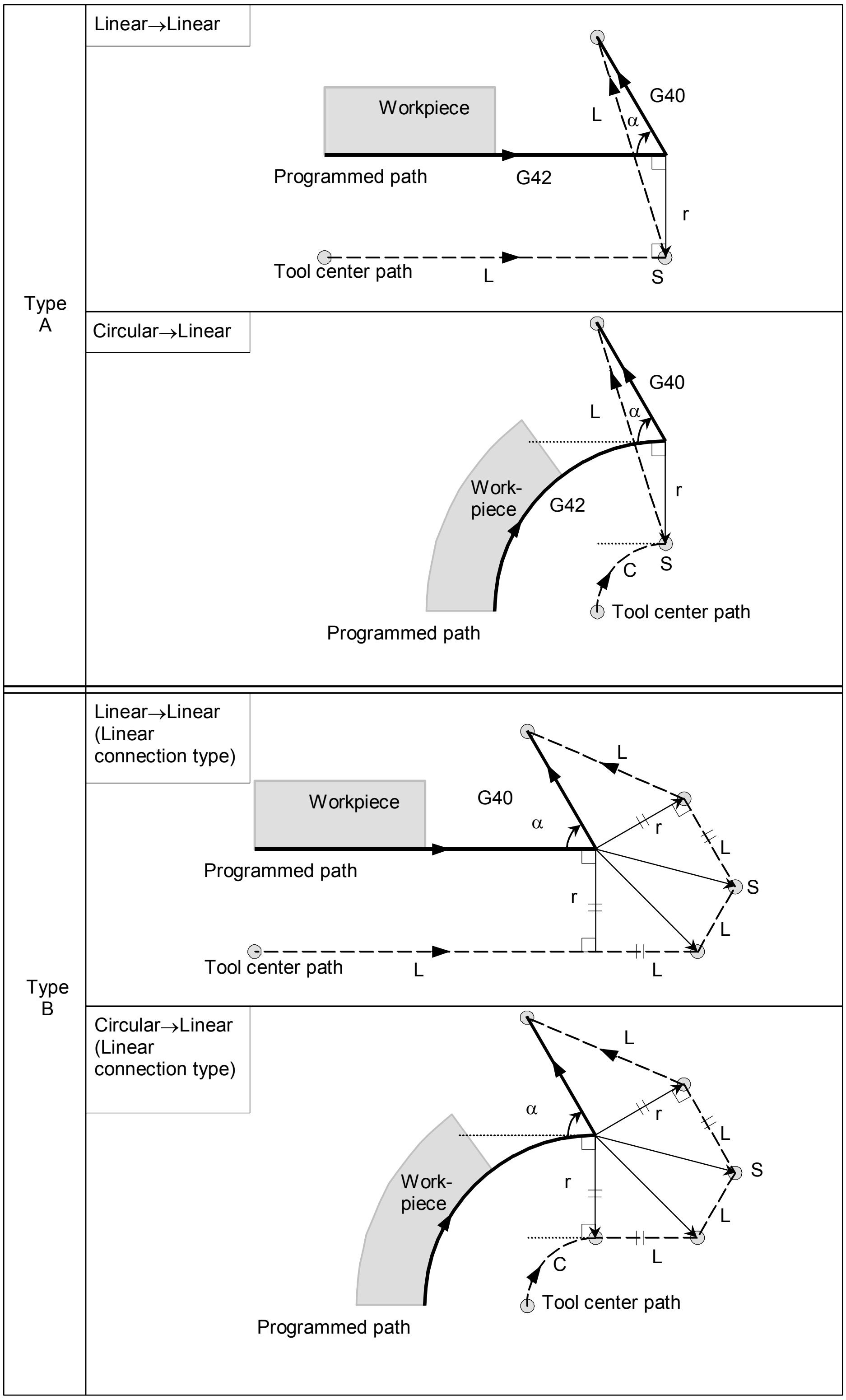
Tool path has two types A and B, and they are selected by parameter SUP (No.5003#0).

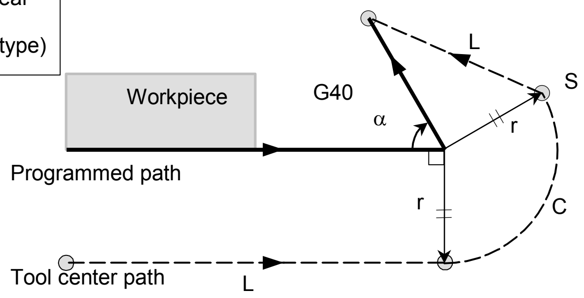
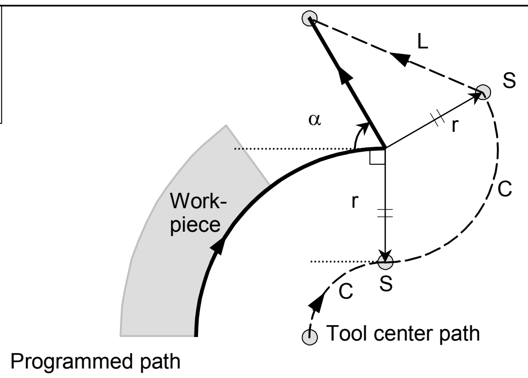


<table>
<tr>
<td rowspan="2">Type
B</td>
<td>Linear→Linear
(Circular
connection type)</td>
<td>
</td>
</tr>
<tr>
<td>Circular→Linear
(Circular
connection type)</td>
<td>
</td>
</tr>
</table>

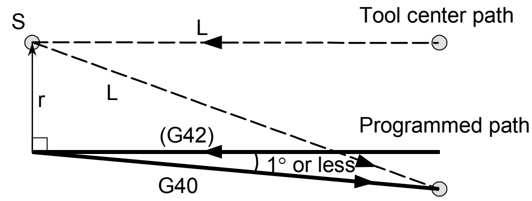
- If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle (α<90°)

Tool path has two types A and B, and they are selected by parameter SUP (No.5003#0).



Type B	Linear→Linear (Circular connection type)	
	Circular→Linear (Circular connection type)	

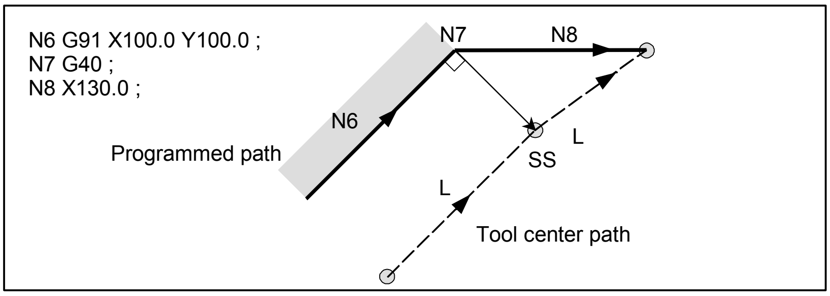
- **If the cancel block is a block with tool movement, and the tool moves around the outside at an acute angle of 1 degree or less in a linear → linear manner ($\alpha \leq 1°$)**



- A block without tool movement specified together with offset cancel

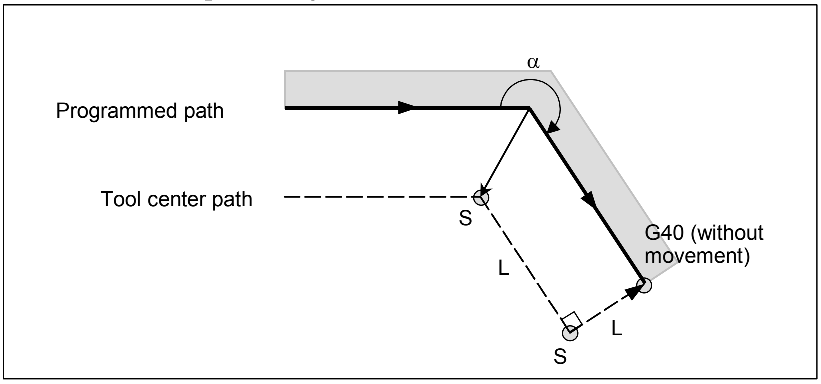
For types A and B

In the block preceding the cancel block, a vector is created with a size equal to the cutter or tool nose radius compensation value in the vertical direction. The tool does not operate in the cancel block. The remaining vectors are canceled with the next move command.



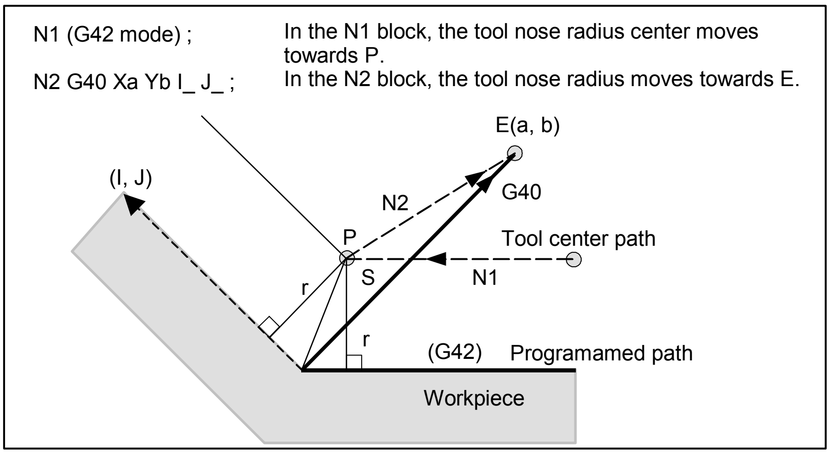
For type C

The tool shifts by the compensation value in the direction vertical to the block preceding the cancel block.

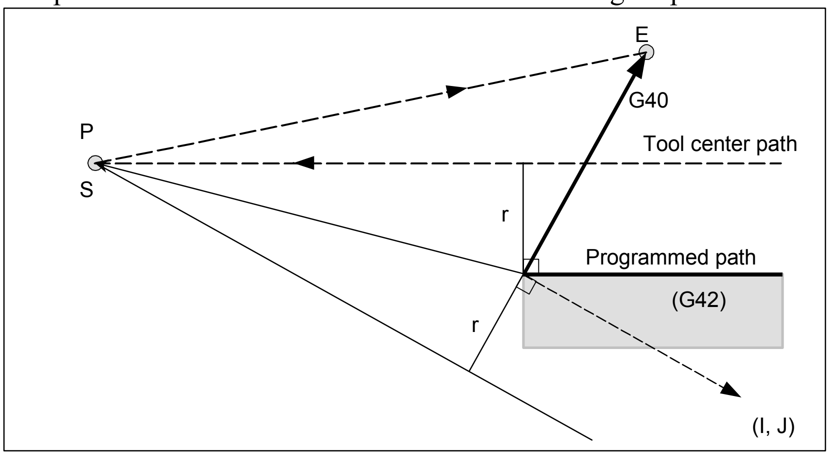


- Block containing G40 and I_J_K_ The previous block contains G41 or G42

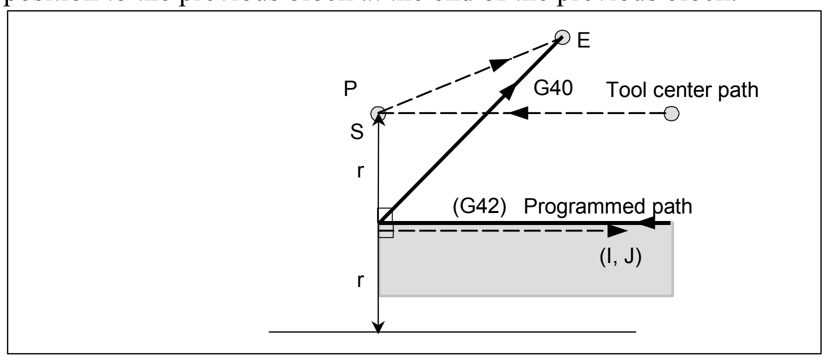
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end point determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified.

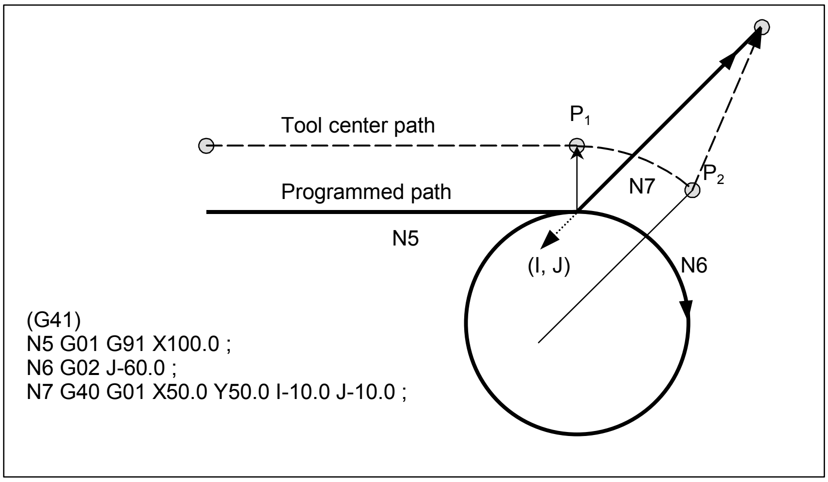


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



- Length of the tool center path larger than the circumference of a circle

In the Example shown below, the tool does not trace the circle more than once. It moves along the arc from P_1 to P_2. The interference check function described below may raise an alarm.
To make the tool trace a circle more than once, program two or more arcs.



6.6.5 Prevention of Overcutting Due to Cutter or Tool Nose Radius Compensation

Explanation

- Machining a groove smaller than the diameter of the tool

Since the cutter compensation forces the tool center path to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.

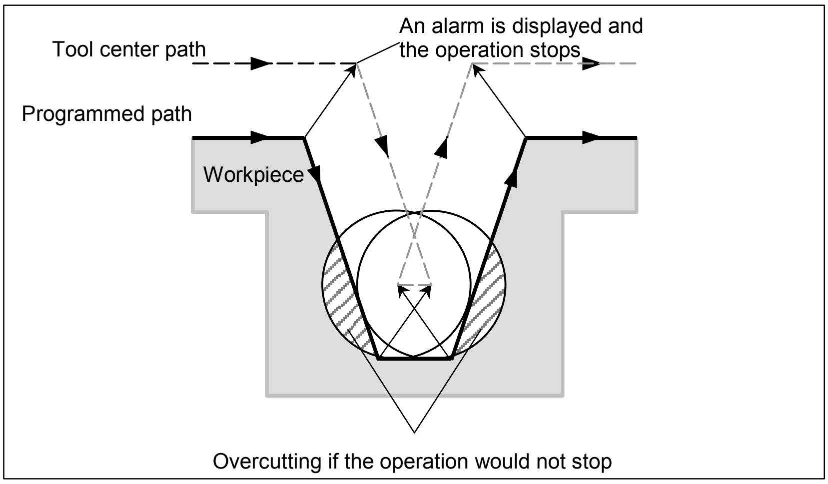


Fig. 6.6.5 (a) Machining a groove smaller than the diameter of the tool

- Machining a step smaller than the tool radius

For a figure in which a workpiece step is specified with an arc, the tool center path will be as shown in Fig. 6.6.5 (b). If the step is smaller than the tool radius, the tool center path usually compensated as shown in Fig. 6.6.5 (c) may be in the direction opposite to the programmed path. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued.

If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.

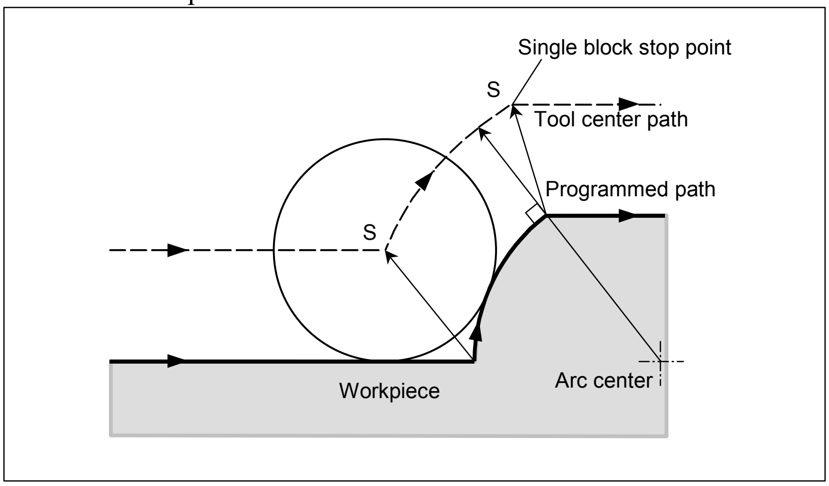


Fig. 6.6.5 (b) Machining a step larger than the tool radius

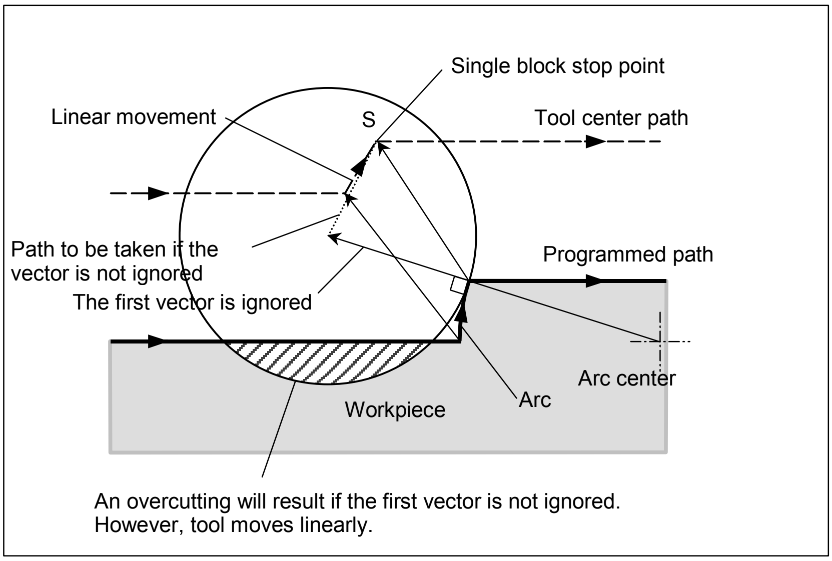
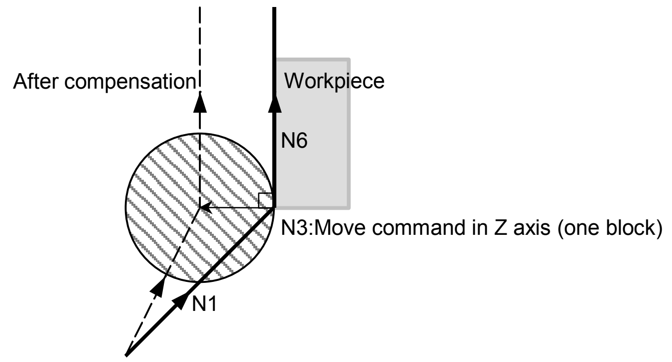


Fig. 6.6.5 (c) Machining a step smaller than the tool radius

- Starting compensation and cutting along the Z-axis

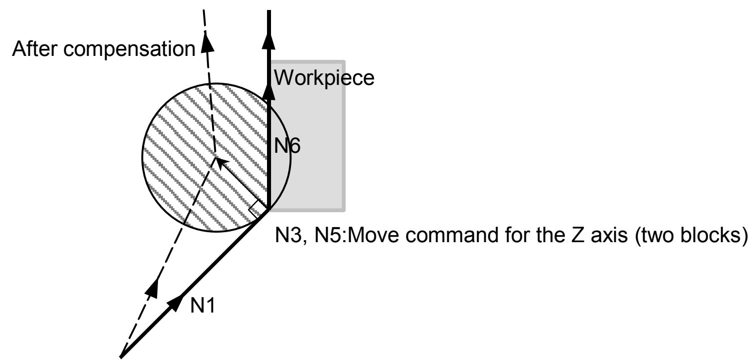
It is usually used such a method that the tool is moved along the Z axis after the cutter compensation (normally XY plane) is effected at some distance from the workpiece at the start of the machining. In the case above, if it is desired to divide the motion along the Z axis into
rapid traverse and cutting feed, follow the procedure below.
Let us consider the following program, assuming the number of blocks to read in cutter compensation mode (parameter (No. 19625)) to be 3.

```
N1 G91 G00 G41 X500.0 Y500.0 D1 ;
N3 G01 Z-300.0 F100 ;
N6 Y1000.0 F200 ;
```



In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure above.
Then, suppose that the block N3 (move command in Z axis) is divided into N3 and N5.

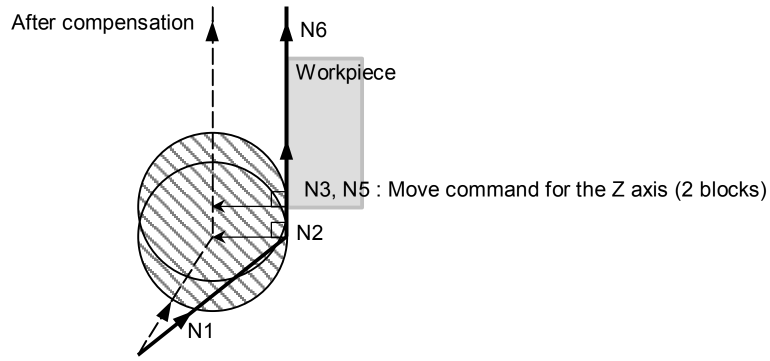
```
N1 G91 G00 G41 X500.0 Y500.0 D1 ;
N3 G01 Z-250.0 ;
N5 G01 Z-50.0 F100 ;
N6 Y1000.0 F200 ;
```



At this time, because the number of blocks to read is 3, blocks up to N5 can be read at the start of N1 compensation, but block N6 cannot be read. As a result, compensation is performed only on the basis of the information in block N1, and a vertical vector is created at the end of the compensation start block. Usually, therefore, overcutting will result as shown in the figure above.

In such a case, it is possible to prevent overcutting by specifying a command with the exactly the same direction as the advance direction immediately before movement along the Z axis beforehand, after the tool is moved along the Z axis using the above rule.

```
N1 G91 G00 G41 X500.0 Y400.0 D1 ;
N2 Y100.0 ;
N3 Z-250.0 ;
N5 G01 Z-50.0 F100 ;
N6 Y1000.0 F200 ;
```



As the block with sequence No. N2 has the move command in the same direction as that of the block with sequence No. N6, the correct compensation is performed.

Alternatively, it is possible to prevent overcutting in the same way by specifying an IJ type vector with the same direction as the advance direction in the start-up block, as in N1 G91 G00 G41 X500. Y500. I0 J1 D1;, after the tool has moved along the Z axis.

6.6.6 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

Explanation

- Condition under which an interference check is possible

To perform an interference check, it is necessary to read at least three blocks with tool movement. If, therefore, three or more blocks with tool movement cannot be read in offset mode because blocks without tool movement, such as independent auxiliary function and dwell, are specified in succession, excessive or insufficient cutting may occur because an interference check fails. Assuming the number of blocks to read in offset mode, which is determined by parameter (No. 19625), to be N and the number of commands in those N blocks without tool movement that have been read to be M, the condition under which an interference check is possible is

$$(N - 3) \geq M.$$

For example, if the maximum number of blocks to read in offset mode is 8, an interference check is possible even if up to five blocks without tool movement are specified. In this case, three adjacent blocks can be checked for interference, but any subsequent interference that may occur cannot be detected.

- Interference check method

Two interference check methods are available, direction check and circular angle check. Parameter CNC (No. 5008#1) and parameter CNV (No. 5008#3) are used to specify whether to enable these methods.

Parameter CNV	Parameter CNC	Operation
0	0	An interference check is enabled, and a direction check and a circular angle check can be performed.
0	1	An interference check is enabled, and only a circular angle check is performed.
1	–	An interference check is disabled.

> **NOTE**
> There are no settings for performing a direction check only.

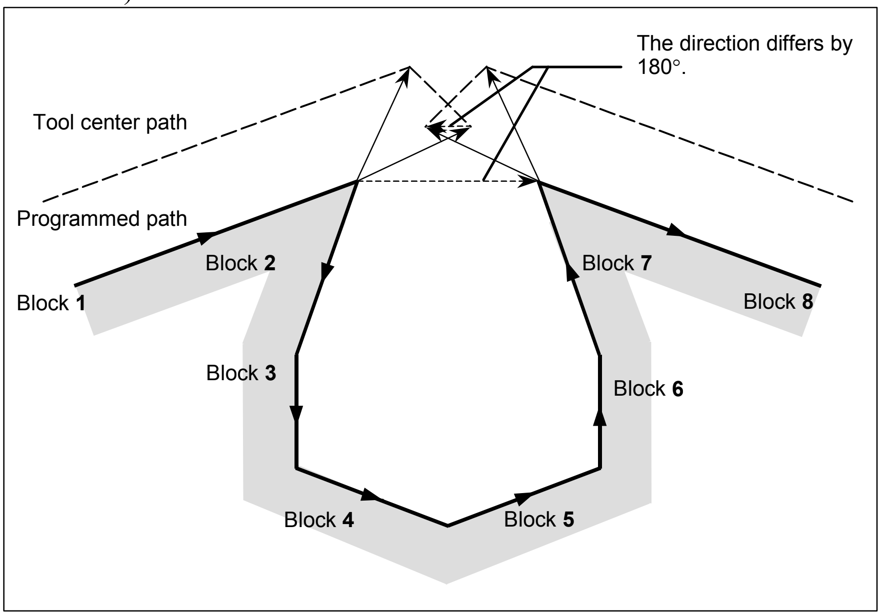
- Interference reference <1> (direction check)

Assuming the number of blocks to read during cutter compensation to be N, a check is first performed on the compensation vector group calculated in (block 1 - block 2) to be output this time and the compensation vector group calculated in (block N-1 - block N); if they intersect, they are judged to interfere. If no interference is found, a check is performed sequentially in the direction toward the compensation vector group to be output this time, as follows:

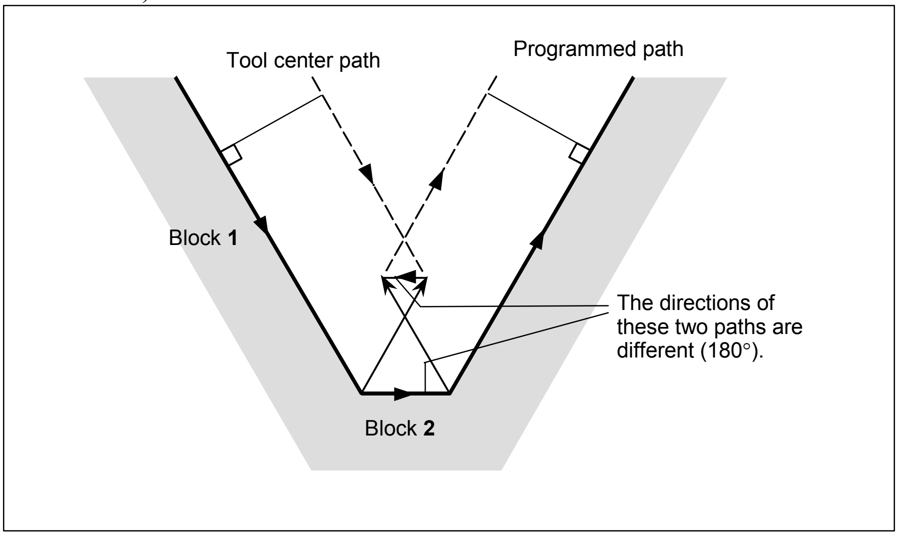
(Block 1 - block 2) and (block N-2 - block N-1)
(Block 1 - block 2) and (block N-3 - block N-2)
:
:
(Block 1 - block 2) and (block 2 - block 3)

Even if multiple number of compensation vector groups are generated, a check is performed on all pairs.
The judgment method is as follows: For a check on the compensation vector group in (block 1 - block 2) and those in (block N-1 - block N), the direction vector from the specified (end point of block 1) to the (end point of block N-1) is compared with the direction vector from the (point resulting from adding the compensation vector to be checked to the end of block 1) to the (point resulting from adding the compensation vector to be checked to the end of block N-1), and if the direction is 90^o or greater or 270^o or less, they are judged to intersect and interfere. This is called a direction check.

Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 7 end-point vector)



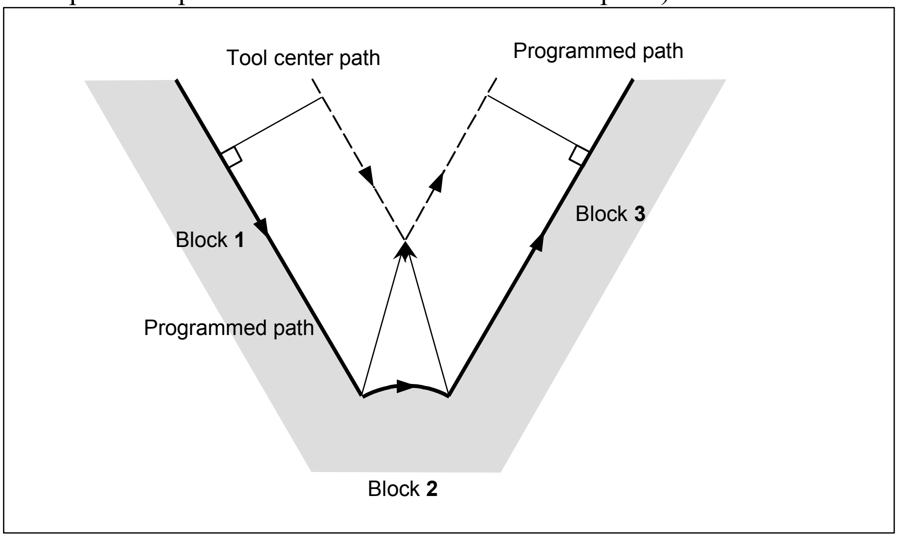
Example of interference standard <1>
(If the block 1 end-point vector intersects with the block 2 end-point vector)



- Interference reference <2> (circular angle check)

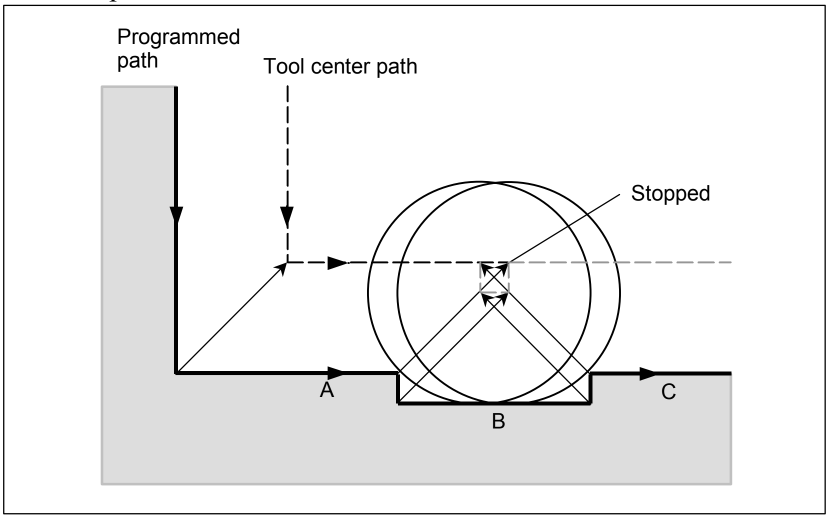
In a check on three adjacent blocks, that is, a check on the compensation vector group calculated on (block 1 - block 2) and the compensation vector group calculated on (block 2 - block 3), if block 2 is circular, a check is performed on the circular angle between the start and end points of the programmed path and the circular angle of the start and end point of the post-compensation path, in addition to direction check <1>. If the difference is 180° or greater, the blocks are judged to interfere. This is called a circular angle check.

Example of <2> (if block 2 is circular and the start point of the post-compensation arc coincide with the end point)



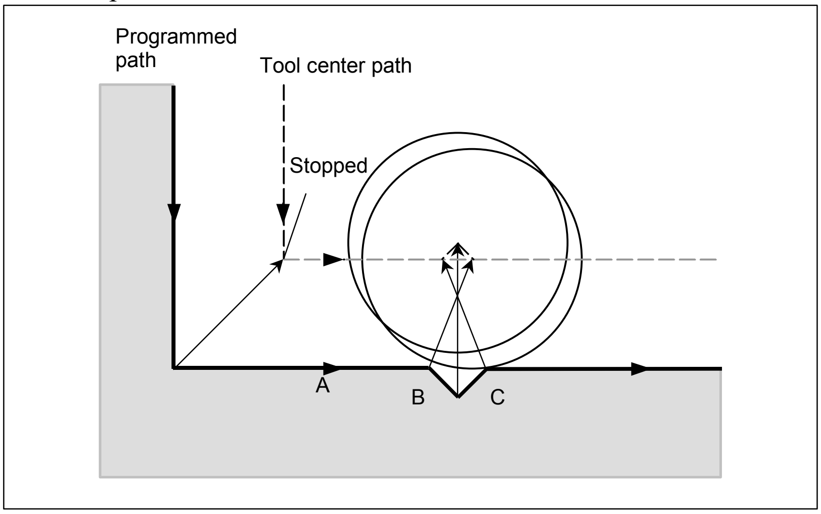
- When interference is assumed although actual interference does not occur

<1> Depression which is smaller than the cutter or tool nose radius compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after the cutter compensation, the tool stops and an alarm is displayed.

<2> Groove which is smaller than the cutter or tool nose radius compensation value



Like <1>, an alarm is displayed because of the interference as the direction is reverse in block B.

6.6.6.1 Operation to be performed if an interference is judged to occur

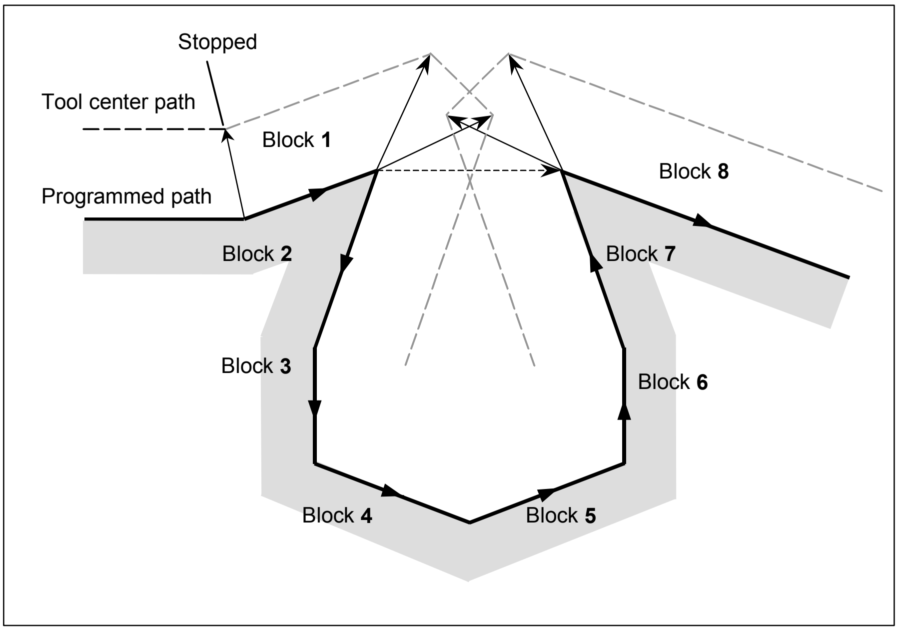
The operation to be performed if an interference check judges that an interference (due to overcutting) occurs can be either of the following two, depending on the setting of parameter CAV (No. 19607#5).

Parameter CAV	Function	Operation
0	Interference check alarm function	An alarm stop occurs before the execution of the block in which overcutting (interference) occurs.
1	Interference check avoidance function	The tool path is changed so that overcutting (interference) does not occur, and processing continues.

6.6.6.2 Interference check alarm function

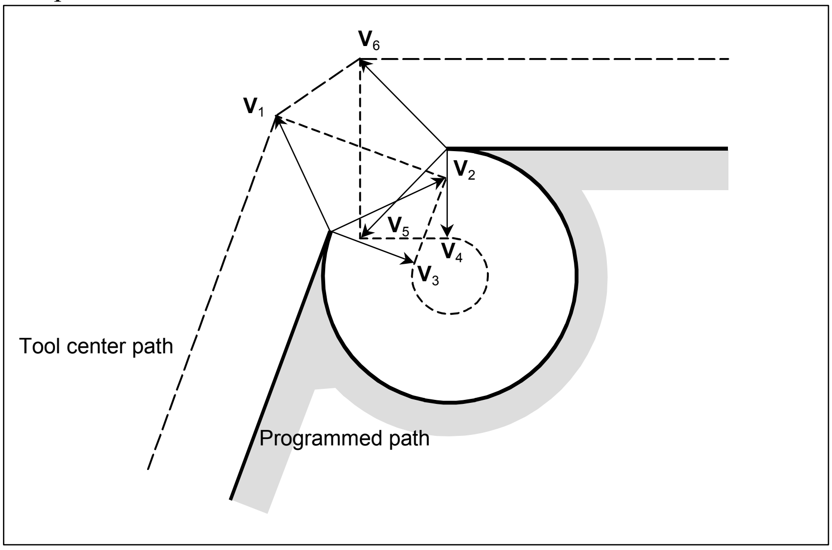
- Interference other than those between adjacent three blocks

If the end-point vector of block 1 and the end-point vector of block 7 are judged to interfere as shown in the figure, an alarm will occur before the execution of block 1 so that the tool stops. In this case, the vectors will not be erased.

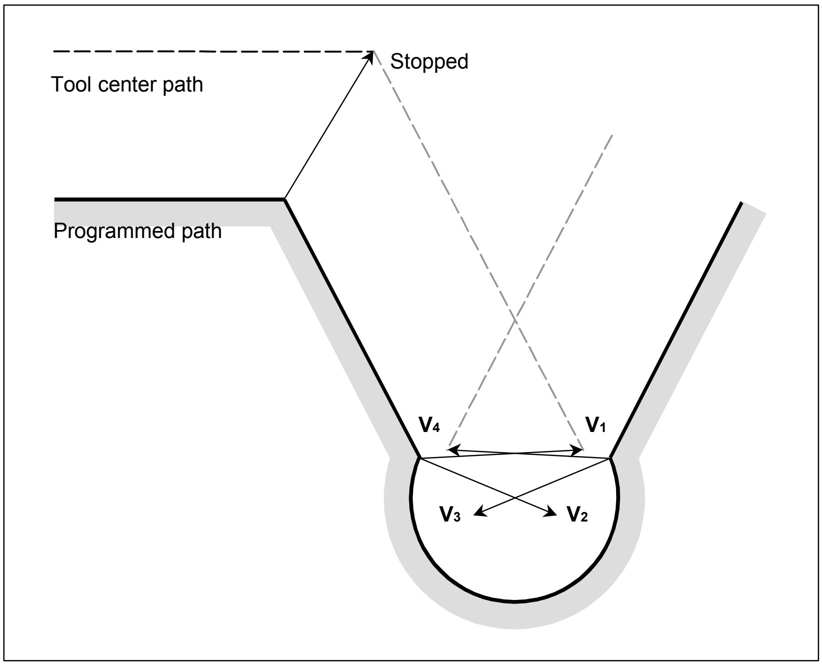


- Interference between adjacent three blocks

If an interference is judged to occur between adjacent three blocks, the interfering vector, as well as any vectors existing inside of it, is erased, and a path is created to connect the remaining vectors. In the Example shown in the figure below, V_2 and V_5 interfere, so that V_2 and V_5 are erased, so are V_3 and V_4, which are inside of them, and V_1 is connected to V_6. The operation during this time is linear interpolation.



If, after vector erasure, the last single vector still interferes, or if there is only one vector at the beginning and it interferes, an alarm will occur immediately after the start of the previous block (end point for a single block) and the tool stops. In the Example shown in the figure below, V_2 and V_3 interfere, but, even after erasure, an alarm will occur because the final vectors V_1 and V_4 interfere.



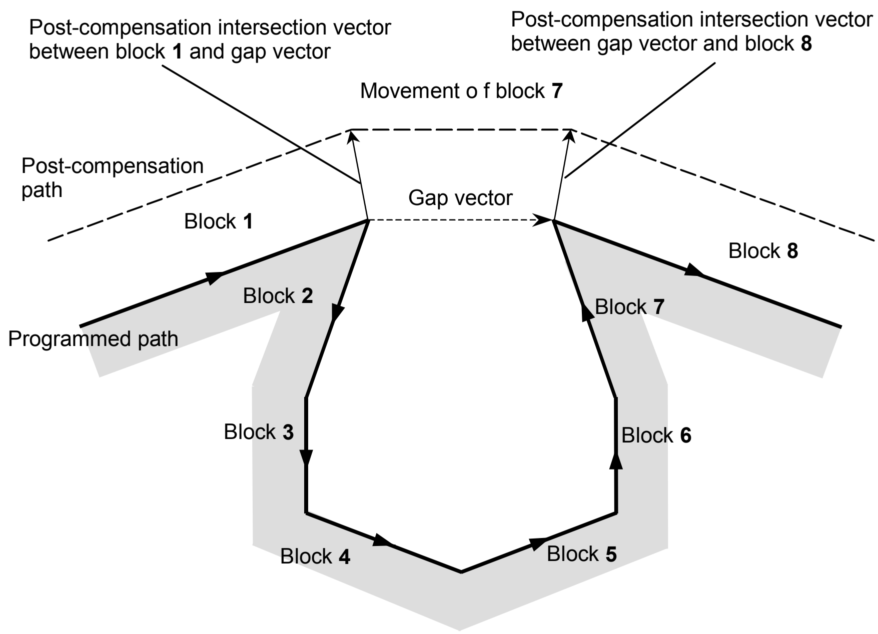
6.6.6.3 Interference check avoidance function

Overview

If a command is specified which satisfies the condition under which the interference check alarm function generates an interference alarm, this function suppresses the generation of the interference alarm, but causes a new compensation vector to be calculated as a path for avoiding interference, thereby continuing machining. For the path for avoiding interference, insufficient cutting occurs in comparison with the programmed path. In addition, depending on the specified figure, no path for avoiding interference can be determined or the path for avoiding interference may be judged dangerous. In such a case, an alarm stop will occur. For this reason, it is not always possible to avoid interference for all commands.

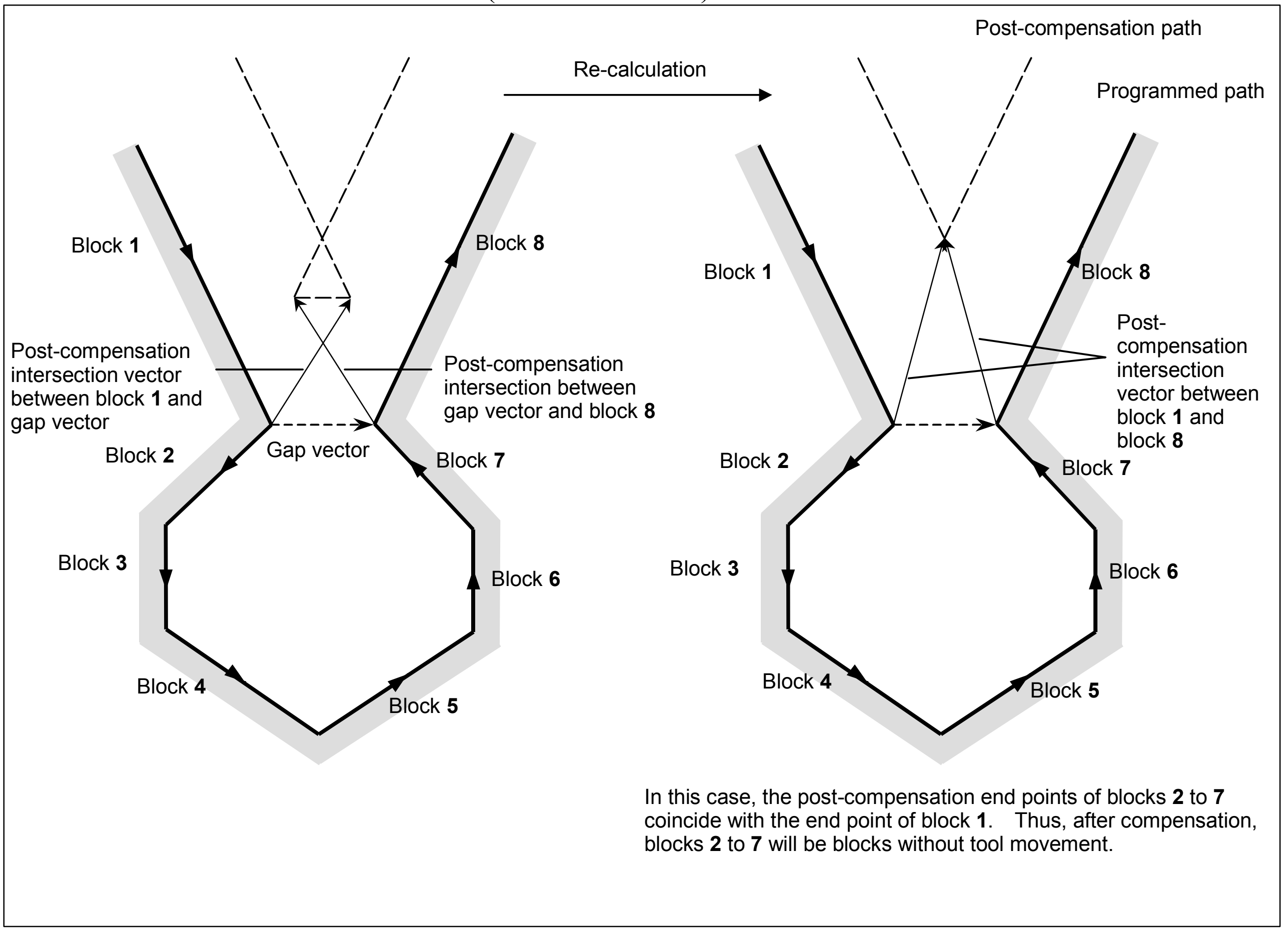
- Interference avoidance method

Let us consider a case in which an interference occurs between the compensation vector between (block 1 - block 2) and the compensation vector between (block N-1 - block N). The direction vector from the end point of block 1 to the end point of block N-1 is called a gap vector. At this time, a post-compensation intersection vector between (block 1 - gap vector) and a post-compensation intersection vector between (gap vector - block N) is determined, and a path connecting them is created.

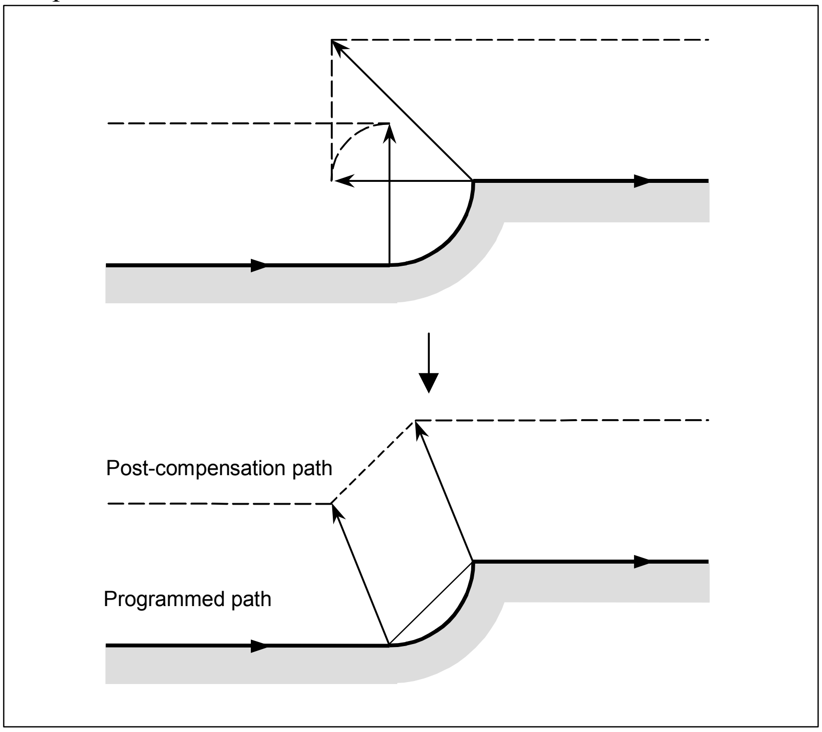


In this case, the post-compensation end points of blocks **2** to **6** coincide with the end point of block **1**. Thus, after compensation, blocks **2** to **6** will be blocks without tool movement.

If the post-compensation intersection vector of (block 1 - gap vector) and the post-compensation intersection vector of (gap vector - block N) further intersect, vector erasure is first performed in the same way as in "Interference between adjacent three blocks". If the last vectors that remains still intersects, the post-compensation intersection vector of (block 1 - block N) is re-calculated.

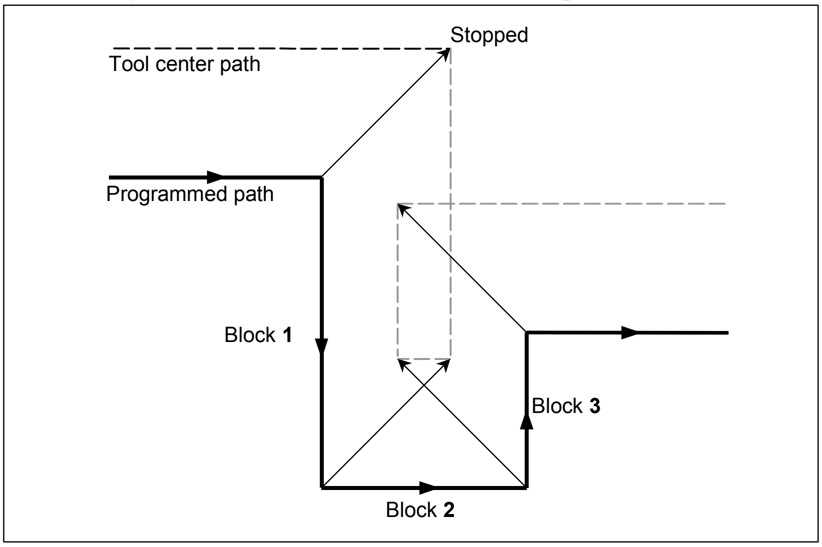


If the cutter or tool nose radius compensation value is greater than the radius of the specified arc as shown in the figure below, and a command is specified which results in compensation with respect to the inside of the arc, interference is avoided by performing intersection calculation with an arc command being assumed a linear one. In this case, avoided vectors are connected with linear interpolation.

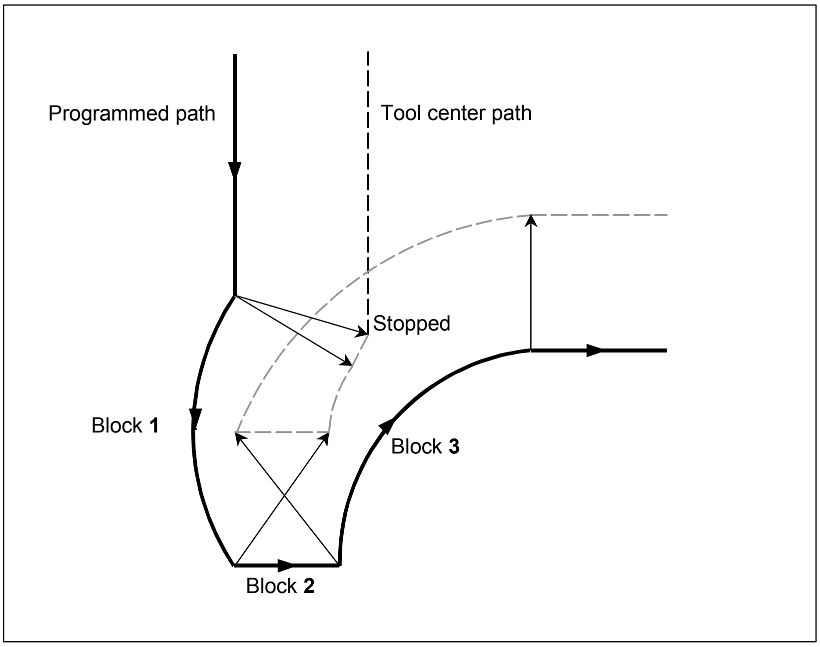


- If no interference avoidance vector exists

If the parallel pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are parallel to each other, no intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop.

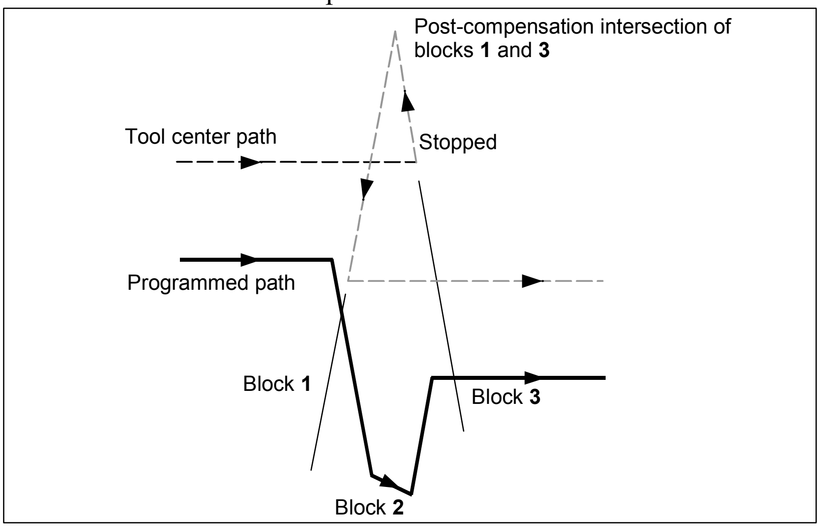


If the circular pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, because blocks 1 and 3 are circular, no post-compensation intersection exists. In this case, an alarm will occur immediately before block 1 and the tool will stop, as in the previous example.



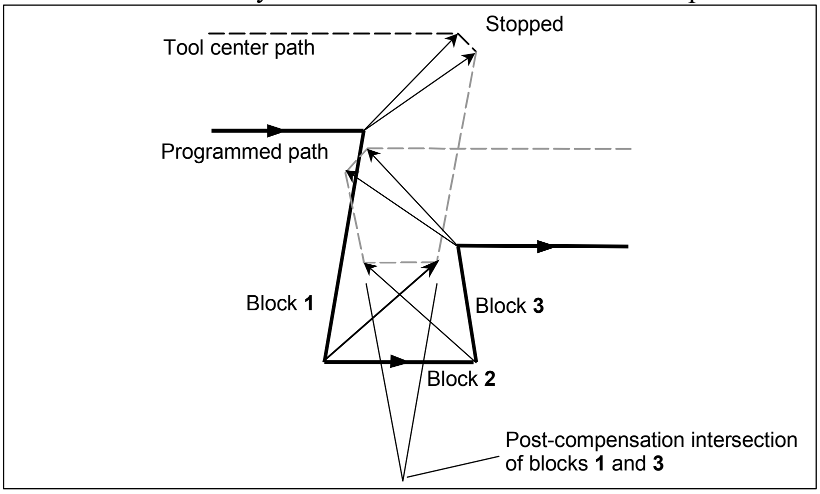
- If it is judged dangerous to avoid interference

If the acute-angle pocket shown in the figure is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the movement direction of the post-avoidance path extremely differs from the previously specified direction. If the post-avoidance path extremely differs from that of the original command (90° or greater or 270° or less), interference avoidance operation is judged dangerous; an alarm will occur immediately before block 1 and the tool will stop.



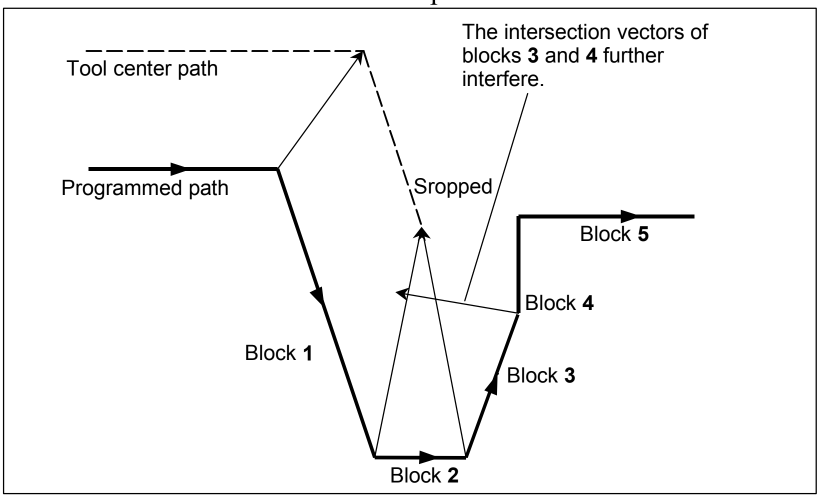
If a pocket in which the bottom is wider than the top, such as that shown in the figure, is to be machined, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, the relation between

blocks 1 and 3 is judged an outer one, the post-avoidance path results in overcutting as compared with the original command. In such a case, interference avoidance operation is judge dangerous; an alarm will occur immediately before block 1 and the tool will stop.



- If further interference with an interference avoidance vector occurs

If the pocket shown in the figure is to be machined, if the number of blocks to read is 3, the end-point vector of block 1 and the end-point vector of block 2 are judged to interfere, and an attempt is made to calculate, as an interference avoidance vector, the intersection vector of the post-compensation path of block 1 and the post-compensation path of block 3. In this case, however, the end-point vector of block 3 that is to be calculated next further interferes with the previous interference avoidance vector. If a further interference occurs to the interference avoidance vector once created and output, the movement in the block will not be performed; an alarm will occur immediately before the block and the tool will stop.



NOTE

1 For "If it is judged dangerous to avoid interference" and "If further interference with an interference avoidance vector occurs", by setting parameter NAA (No. 19607#6) appropriately, it is possible to suppress an alarm to continue machining. For "If no interference avoidance vector exists", however, it is not possible to avoid an alarm regardless of the setting of this parameter.

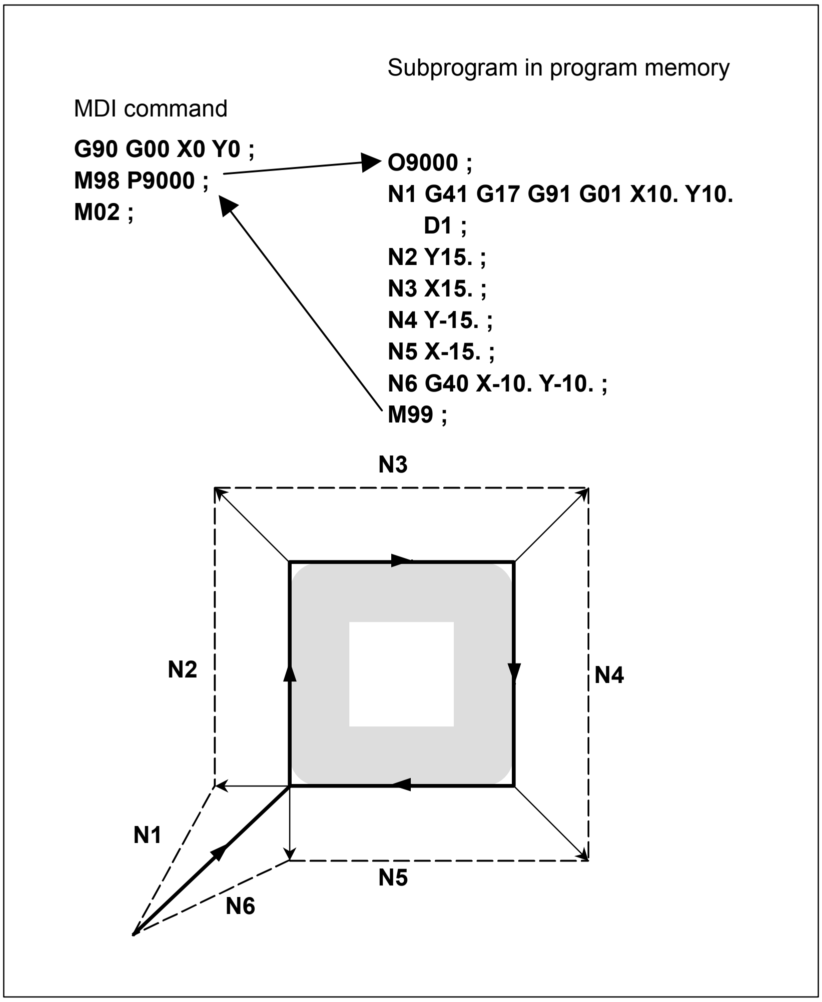
2 If a single block stop occurs during interference avoidance operation, and an operation is performed which differs from the original movement, such as manual intervention, MDI intervention, cutter or tool nose radius compensation value change, intersection calculation is performed with a new path. If such an operation is performed, therefore, an interference may occur again although interference avoidance has been performed once.

6.6.7 Cutter or Tool Nose Radius Compensation for Input from MDI

Explanation

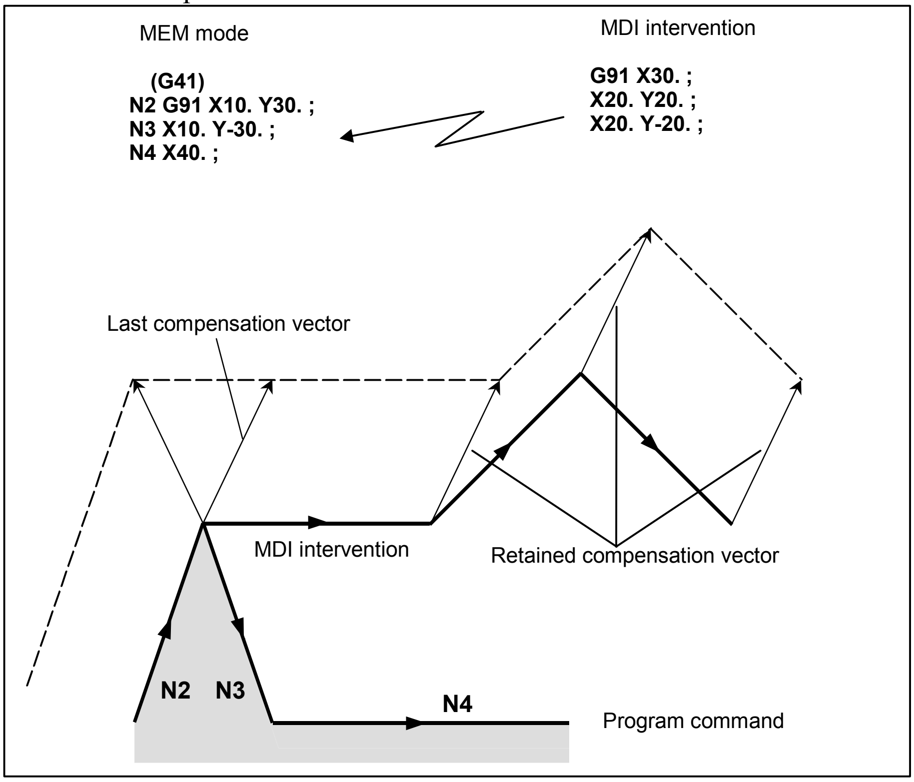
- MDI operation

During MDI operation, that is, if a program command is specified in MDI mode in the reset state to make a cycle start, intersection calculation is performed for compensation in the same way as in memory operation/DNC operation. Compensation is performed in the same way if a subprogram is called from program memory due to MDI operation.



- MDI intervention

If MDI intervention is performed, that is, if a single block stop is performed to enter the automatic operation stop state in the middle of memory operation, DNC operation, and the like, and a program command is specified in MDI mode to make a cycle start, cutter compensation does not perform intersection calculation, retaining the last compensation vector before the intervention.



6.7 VECTOR RETENTION (G38)

In cutter or tool nose radius compensation, by specifying G38 in offset mode, it is possible to retain the compensation vector at the end point of the previous block, without performing intersection calculation.

Format

(In offset mode)
G38 IP_ ;

IP: Value specified for axial movement

Explanation

- Vector retention

By specifying the above command, a vector is created at the end point of the block immediately preceding the G38 block, vertical to that block. In the G38 block, the vertical vector output in the previous block is retained. G38 is a one-shot G code. With the next move command without a G38 command, the compensation vector is re-created.

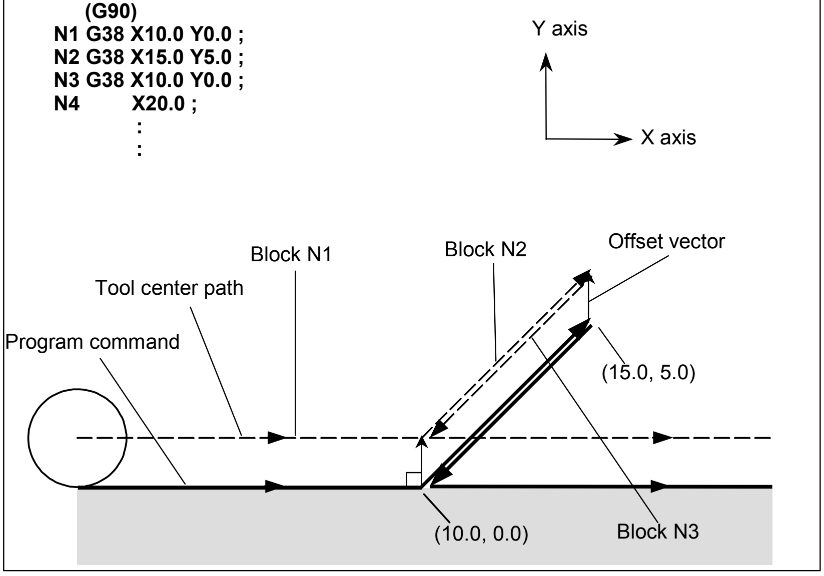
Limitation

- Mode

Specify G38 in either G00 or G01 mode. If it is specified in G02 or G03 (circular interpolation) mode, a radial error may occur at the start and end points.

Example

```
        :
        :      (In offset mode)
   (G90)
N1 G38 X10.0 Y0.0 ;
N2 G38 X15.0 Y5.0 ;
N3 G38 X10.0 Y0.0 ;
N4     X20.0 ;
        :
        :
```



6.8 CORNER CIRCULAR INTERPOLATION (G39)

By specifying G39 in offset mode during cutter or tool nose radius compensation, corner circular interpolation can be performed. The radius of the corner circular interpolation equals the compensation value.

Format

In offset mode
G39 ;
or

$$\textbf{G39}\begin{Bmatrix}\textbf{I\_ J\_}\\ \textbf{I\_ K\_}\\ \textbf{J\_ K\_}\end{Bmatrix};$$

Explanation

- Corner circular interpolation

When the command indicated above is specified, corner circular interpolation in which the radius equals compensation value can be performed. G41 or G42 preceding the command determines whether the arc is clockwise or counterclockwise. G39 is a one-shot G code.

- G39 without I, J, or K

When G39; is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block.

- G39 with I, J, and K

When G39 is specified with I, J, and K, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the vector defined by the I, J, and K values.

Limitation

- Move command

In a block containing G39, no move command can be specified. Otherwise, an alarm will occur.

- Inner corner

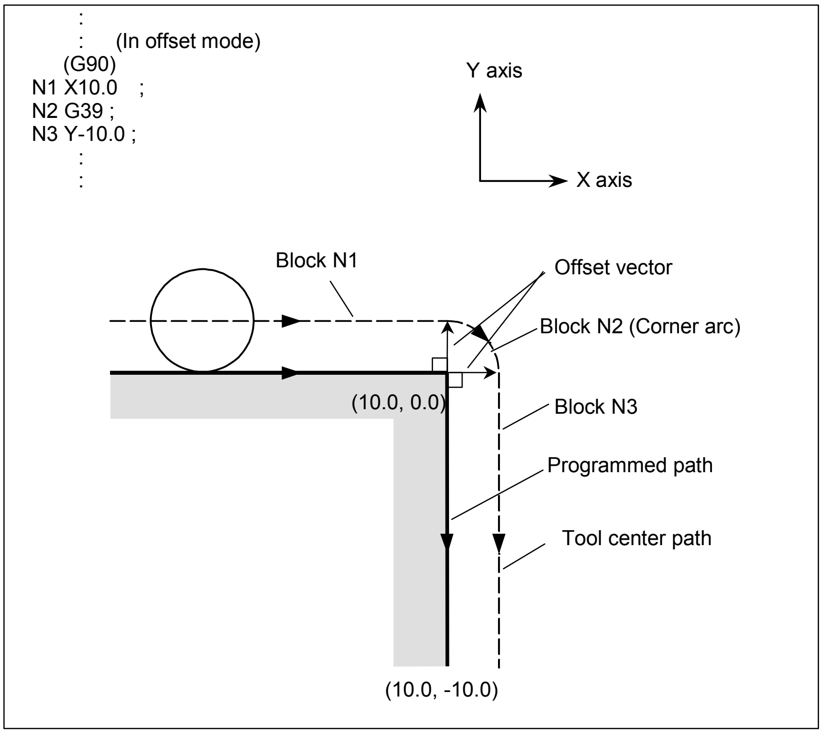
In an inner corner block, G39 cannot be specified. Otherwise, overcutting will occur.

- Corner arc velocity

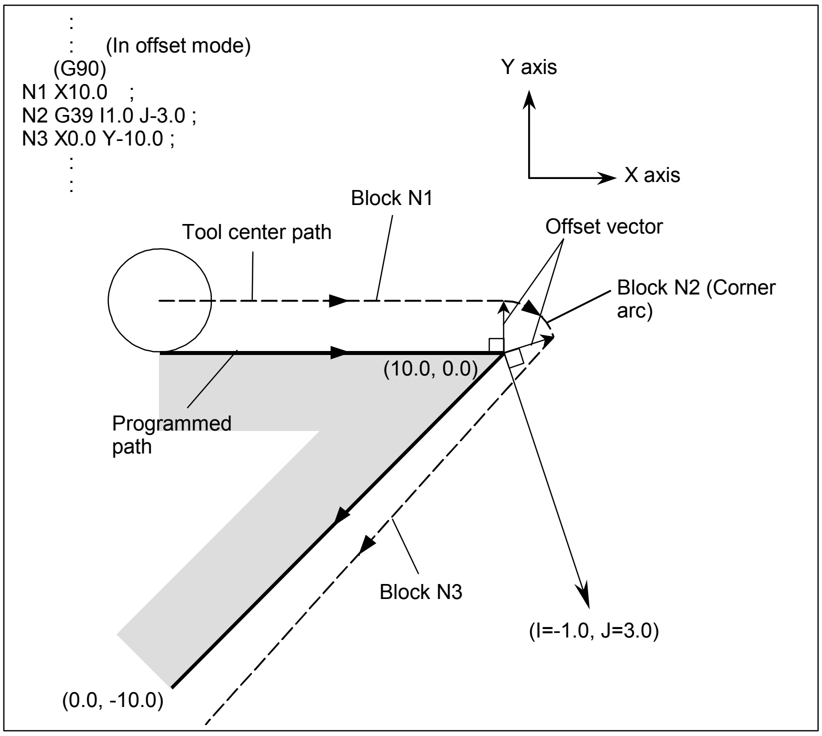
If a corner arc is specified with G39 in G00 mode, the corner arc block velocity will be that of the F command previously specified. If G39 is specified in a state in which no F command has never been specified, the velocity of the corner arc block will be that specified with parameter (No. 1411).

Example

- G39 without I, J, or K



- G39 with I, J, and K



6.9 THREE-DIMENSIONAL CUTTER COMPENSATION (G40, G41)

In cutter compensation C, two-dimensional offsetting is performed for a selected plane. In three-dimensional cutter compensation, the tool can be shifted three-dimensionally when a three-dimensional offset direction is programmed.

Format

- Start up (Starting three-dimensional cutter compensation)

When the following command is executed in the cutter compensation cancel mode, the three-dimensional cutter compensation mode is set:

G41 Xp_Yp_Zp_ I_ J_ K_D_ ;

Xp : X-axis or a parallel axis
Yp : Y-axis or a parallel axis
Zp : Z-axis or a parallel axis
I, J, K : See "Explanation".
D : Code for specifying as the cutter compensation value (1-3 digits) (D code)

- Canceling three-dimensional cutter compensation

When the following command is executed in the three-dimensional cutter compensation mode, the cutter compensation cancel mode is set:

- When canceling the three-dimensional cutter compensation mode and tool movement at the same time

  G40 Xp_Yp_Zp_ ;
  or
  Xp_Yp_Zp_ D00 ;

- When only canceling the vector

  G40;
  or
  D00;

- Selecting offset space

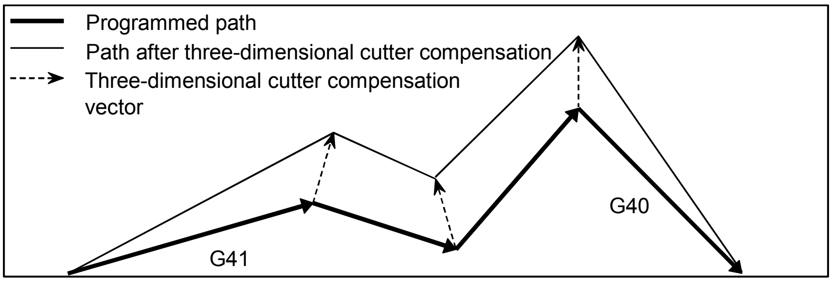
The three-dimensional space where three-dimensional cutter compensation is to be executed is determined by the axis addresses specified in the startup block containing the G41 command. If Xp, Yp, or Zp is omitted, the corresponding axis, X-, Y-, or Z-axis (the basic three axis), is assumed.
(Example)
When the U-axis is parallel to the X-axis, the V-axis is parallel to the Y-axis, and the W-axis is parallel to the Z-axis

G41 X_I_J_K_D_;	XYZ space
G41 U_V_Z_I_J_K_D_;	UVZ space
G41 W_I_J_K_D_;	XYW space

Explanation

- Three-dimensional cutter compensation vector

In three-dimensional cutter compensation mode, the following three-dimensional cutter compensation vector is generated at the end of each block:



The three-dimensional cutter compensation vector is obtained from the following expressions:

$$Vx = \frac{i \times r}{p} \quad \text{(Vector component along the Xp-axis)}$$

$$Vy = \frac{j \times r}{p} \quad \text{(Vector component along the Yp-axis)}$$

$$Vz = \frac{k \times r}{p} \quad \text{(Vector component along the Zp-axis)}$$

In the above expressions, i, j, and k are the values specified in addresses I, J, and K in the block. r is the offset value corresponding to the specified offset number. p is the value obtained from the following expression:

$$p = \sqrt{i^2 + j^2 + k^2}$$

When the user wants to program the magnitude of a three-dimensional cutter compensation vector as well as its direction, the value of p in the expressions of Vx, Vy, and Vz can be set as a constant in parameter (No. 5011.)
If the parameter is set to 0, however, p is determined as follows:

$$p = \sqrt{i^2 + j^2 + k^2}$$

- Relationship between three-dimensional cutter compensation and other compensation functions

Tool length compensation	The specified path is shifted by three-dimensional cutter compensation and the subsequent path is shifted by tool length compensation.
Tool offset	When tool offset is specified in the three-dimensional cutter compensation mode, an alarm is issued (alarm PS0042).
Cutter compensation	When addresses I, J, and K are all specified at startup, three-dimensional cutter compensation mode is set. When not all of the addresses are specified, cutter compensation C mode is set. Therefore, cutter compensation C cannot be specified in three-dimensional cutter compensation mode and three-dimensional cutter compensation cannot be specified in cutter compensation C mode.

- Specifying I, J, and K

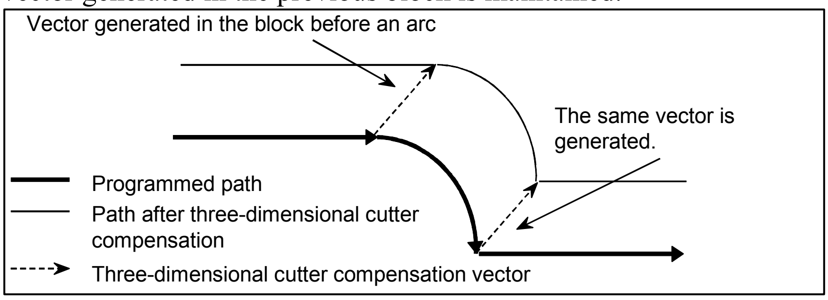
Addresses I, J, and K must all be specified to start three-dimensional cutter compensation. When even one of the three addresses is omitted, two-dimensional cutter compensation C is activated. When a block specified in three-dimensional cutter compensation mode contains none of addresses I, J, and K, the same vector as the vector generated in the previous block is generated at the end of the block.

- G42

Generally, G41 is specified to start three-dimensional cutter compensation. Instead of G41, G42 can be specified for startup. With G42, three-dimensional cutter compensation is performed in the opposite direction.

- Offset vector in interpolation

When circular interpolation, helical interpolation (both specified with G02, G03), or involute interpolation (G02.2, G03.2) is specified, the vector generated in the previous block is maintained.



- Reference position return check (G27)

Before specifying reference position return check (G27), cancel three-dimensional cutter compensation. In the compensation mode, G27 brings the tool to a position shifted by the offset value. If the position the tool reached is not the reference position, the reference position return LED does not go on (the alarm PS0092 alarm is issued).

- Return to a reference position (G28, G30, G30.1)

When return to the reference position (G28), to the second, third, or fourth reference position (G30), or to the floating reference position (G30.1) is specified, the vector is cleared at a middle point.

- Alarm issued at startup

If one of the following conditions is present at the startup of three-dimensional cutter compensation, an alarm is issued:

- Two or more axes are specified in the same direction. (alarm PS0047)
- Although Xp, Yp, or Zp is omitted, the basic three axes are not set. (alarm PS0048)

- Alarm during three-dimensional cutter compensation

If one of the following G codes is specified in the three-dimensional cutter compensation mode, an alarm is issued:

G05 High-speed cycle machining (alarm PS0178)
G31 Skip function (alarm PS0036)
G51 Scaling (alarm PS0141)

- Commands that clear the vector

When one of the following G codes is specified in three-dimensional cutter compensation mode, the vector is cleared:

G73 Peck drilling cycle
G74 Reverse tapping cycle
G76 Fine boring
G80 Canned cycle cancel
G81 Drilling cycle, spot boring
G82 Drilling cycle, counterboring
G83 Peck drilling cycle
G84 Tapping cycle
G85 Boring cycle
G86 Boring cycle
G87 Back boring cycle
G88 Boring cycle
G89 Boring cycle
G53 Machine coordinate system selection

- Commands that generate the same vector as the vector in the previous block

When one of the following G codes is specified in three-dimensional cutter compensation mode, the same vector as the vector generated in the previous block is generated at the end point of the next movement:

G02 Circular or helical interpolation (CW)
G03 Circular or helical interpolation (CCW)
G02.2 Involute interpolation (CW)
G03.2 Involute interpolation (CCW)
G04 Dwell
G10 Data setting
G22 Stored stroke check function enabled

6.10 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values include tool geometry compensation values and tool wear compensation (Fig. 6.10 (a)).

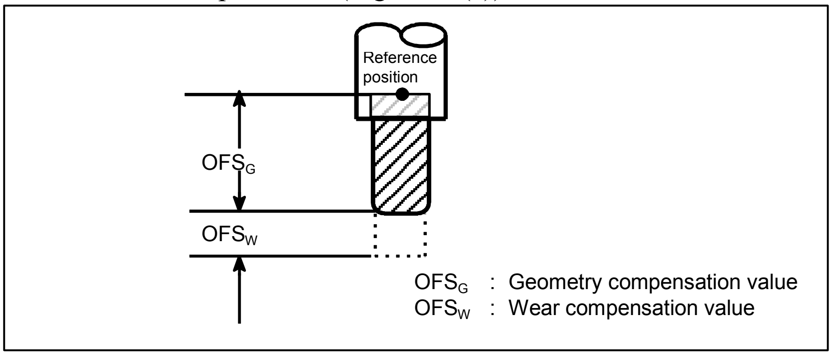


Fig. 6.10 (a) Geometric compensation and wear compensation

Tool compensation values can be entered into CNC memory from the MDI panel (see section III-11.1.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, cutter compensation, or the tool offset.
Three types of tool compensation memories are available according to the compensation value configuration: tool compensation memory A, B, and C. One of the types can be selected.

Explanation

- Tool compensation memory A

In tool compensation memory A, memory for geometry compensation and memory for wear compensation are not distinguished from each other. So, the sum of geometry compensation and wear compensation values is to be set in the compensation memory. Moreover, no distinction is made between memory for cutter compensation (for D code) and memory for tool length compensation (for H code).

Setting example

Compensation number	Compensation value (geometry+wear)	Common to D code/H code
001	10.000	For D code
002	20.000	For D code
003	100.000	For H code
:	:	:

- Tool compensation memory B

In tool compensation memory B, memory for geometry compensation and memory for wear compensation are prepared separately. So, geometry compensation values and wear compensation values can be set separately. However, no distinction is made between memory for cutter compensation (for D code) and memory for tool length compensation (for H code).

Setting example

Compensation number	For geometry compensation	For wear compensation	Common to D code/H code
001	10.100	0.100	For D code
002	20.200	0.200	For D code
003	100.000	0.100	For H code
:	:	:	:

- Tool compensation memory C

In tool compensation memory C, memory for geometry compensation and memory for wear compensation are prepared separately. So, geometry compensation values and wear compensation values can be set separately. Moreover, memory for cutter compensation (for D code) and memory for tool length compensation (for H code) are prepared separately.

Setting example

Compensation number	D code		H code	
	For geometry compensation	For wear compensation	For geometry compensation	For wear compensation
001	10.000	0.100	100.000	0.100
002	20.000	0.200	200.000	0.300
:	:	:	:	:

- Unit and valid range of tool compensation values

A unit and valid range of tool offset values can be selected from the following by parameter setting:

Unit and valid range of tool compensation values (metric input)

OFE	OFD	OFC	OFA	Unit	Valid range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

Unit and valid range of tool compensation values (inch input)

OFE	OFD	OFC	OFA	Unit	Valid range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

- Number of tool compensation data items

The number of tool compensation data items used by the entire system varies from one machine to another. Refer to the relevant manual of the machine tool builder.

Format

The format for programming depends on the type of tool compensation memory.

For tool compensation memory A

```
G10 L11 P_ R_ Q_ ;
   P_  :  Tool compensation number
   R_  :  Tool compensation value
   Q_  :  Imaginary tool nose number
```

For tool compensation memory B

```
G10 L_ P_ R_ Q_ ;
   L_  :  Type of compensation memory
          L10 :  Geometry compensation value
          L11 :  Wear compensation value
   P_  :  Tool compensation number
   R_  :  Tool compensation value
   Q_  :  Imaginary tool nose number
```

For tool compensation memory C

```
G10 L_ P_ R_ Q_ ;
   L_  :  Type of compensation memory
          L10 :  Geometry compensation value
                 corresponding to an H code
          L11 :  Wear compensation value
                 corresponding to an H code
          L12 :  Geometry compensation value
                 corresponding to a D code
          L13 :  Wear compensation corresponding
                 to a D code
   P_  :  Tool compensation number
   R_  :  Tool compensation value
   Q_  :  Imaginary tool nose number
```

By specifying G10, a tool compensation value can be set or modified.
When G10 is specified by absolute input (G90), the specified value is used as the new tool compensation value.
When incremental input (G91) is used, a specified value added to the tool compensation value currently set is used as the new tool compensation value.

NOTE
1. Address R follows the increment system for tool offset values.
2. If L is omitted for compatibility with the conventional CNC format, or L1 is specified, the same operation as when L11 is specified is performed.
3. Set a imaginary tool nose number when the cutter compensation function is specified and a imaginary tool nose direction is used.

6.11 COORDINATE SYSTEM ROTATION (G68, G69)

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine. Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.

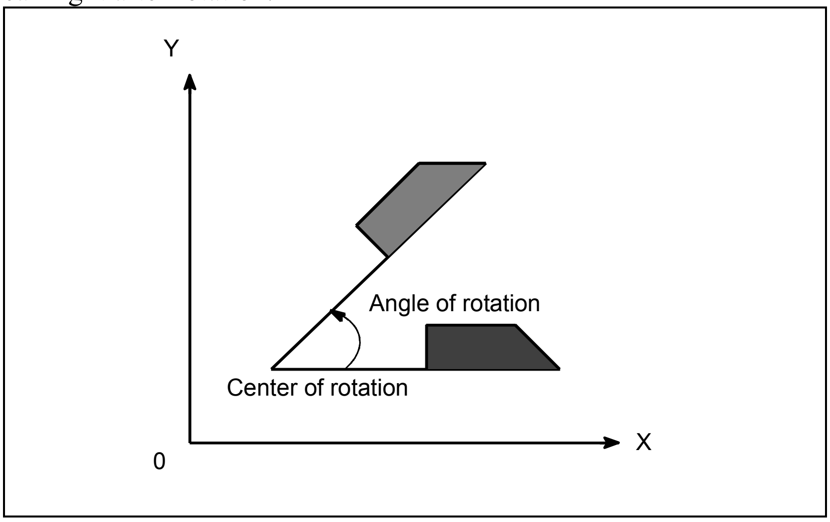


Fig. 6.11 (a) Coordinate system rotation

Format

Format	
{G17 / G18 / G19} G68α68α_β_	Start rotation of a coordinate system.
:	Coordinate system rotation mode (The coordinate system is rotated.)
G69 ;	Coordinate system rotation cancel command
Meaning of command	
G17 (G18 or G19) :	Select the plane in which contains the figure to be rotated.
α_β_	Absolute programming for two of the X_, Y_, and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68
R_	Angular displacement with a positive value indicates counter clockwise rotation. Bit 0 of parameter No. 5400 selects whether the specified angular displacement is always considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).
Least input increment :	0.001 deg
Valid data range :	-360,000 to 360,000

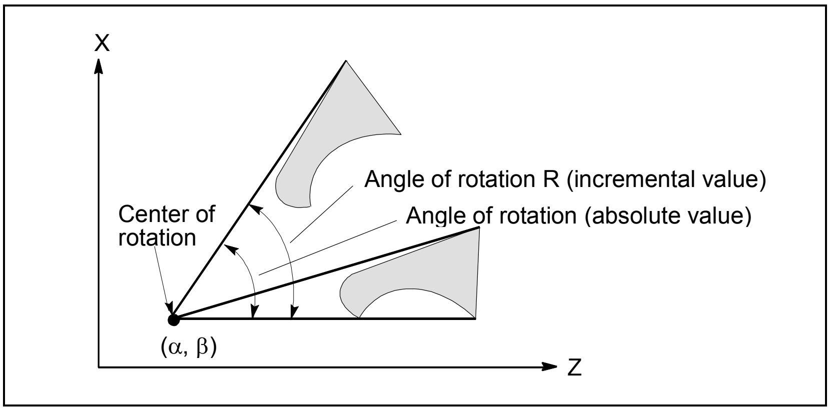


Fig. 6.11 (b) Coordinate system rotation

> **NOTE**
> When a decimal fraction is used to specify angular displacement (R_), the 1's digit corresponds to degree units.

Explanation

- G code for selecting a plane: G17,G18 or G19

The G code for selecting a plane (G17,G18,or G19) can be specified before the block containing the G code for coordinate system rotation (G68). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.

- Incremental programming in coordinate system rotation mode

The center of rotation for an incremental programming programmed after G68 but before an absolute programming is the tool position when G68 was programmed (Fig. 6.11 (c)).

- Center of rotation

When α_β_ is not programmed, the tool position when G68 was programmed is assumed as the center of rotation.

- Angular displacement

When R_ is not specified, the value specified in parameter 5410 is assumed as the angular displacement.

- Coordinate system rotation cancel command

The G code used to cancel coordinate system rotation (G69) may be specified in a block in which another command is specified.

- Tool compensation

Cutter or tool nose radius compensation, tool length compensation, tool offset, and other compensation operations are executed after the coordinate system is rotated.

- Relationship with three-dimensional coordinate conversion (G68, G69)

Both coordinate system rotation and three-dimensional coordinate conversion use the same G codes: G68 and G69. The G code with I, J, and K is processed as a command for three-dimensional coordinate conversion. The G code without I, J, and K is processed as a command for two-dimensional coordinate system rotation.

Limitation

- Commands related to reference position return and the coordinate system

In coordinate system rotation mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode.

- Incremental programming

The first move command after the coordinate system rotation cancel command (G69) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.

Explanation

- Absolute/Incremental position commands

```
N1 G92 X*5000 Y*5000 G69 G17 ;
N2 G68 X7000 Y3000 R60000 ;
N3 G90 G01 X0 Y0 F200 ;
   (G91X5000Y5000)
N4 G91 X10000 ;
N5 G02 Y10000 R10000 ;
N6 G03 X*10000 I*5000 J*5000 ;
N7 G01 Y*10000 ;
N8 G69 G90 X*5000 Y*5000 M02 ;
```

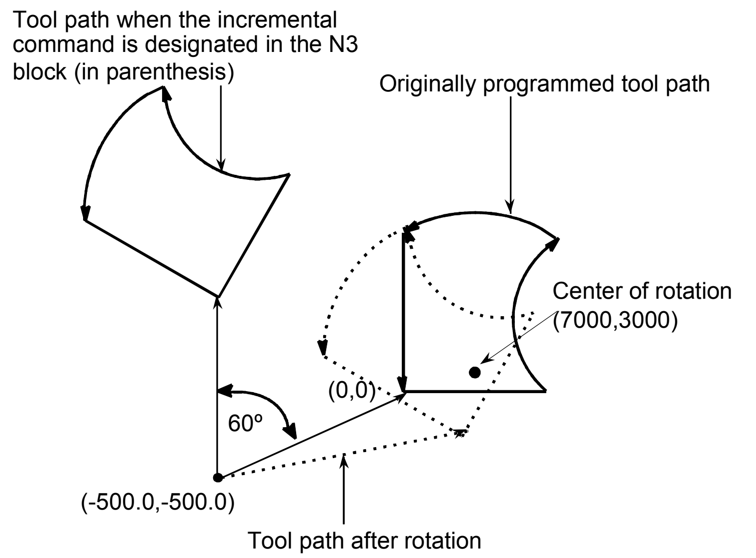


Fig. 6.11 (c) Absolute/incremental programming during coordinate system rotation

- Cutter compensation and coordinate system rotation

It is possible to specify G68 and G69 in cutter compensation mode.
The rotation plane must coincide with the plane of cutter compensation.

```
N1 G92 X0 Y0 G69 G01 ;
N2 G42 G90 X1000 Y1000 F1000 D01 ;
N3 G68 R*30000 ;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500 ;
N6 G01 X*2000 ;
N7 Y*1000 ;
N8 G69 G40 G90 X0 Y0 M30 ;
```

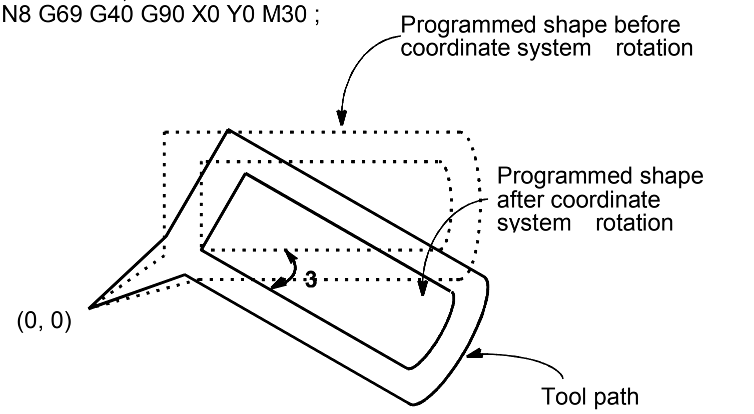


Fig. 6.11 (d) Cutter compensation and coordinate system rotation

- Scaling and coordinate system rotation

If a coordinate system rotation command is executed in the scaling mode (G51 mode), the coordinate value (a,b,) of the rotation center will also be scaled, but not the rotation angle (R). When a move command is issued, the scaling is applied first and then the coordinates are rotated.
A coordinate system rotation command (G68) should not be issued in cutter compensation mode (G41, G42) on scaling mode (G51). The coordinate system rotation command should always be specified prior to setting the cutter compensation mode.

1. When the system is not in cutter compensation mode, specify the commands in the following order :
   G51 ; Scaling mode start
   G68 ; Coordinate system rotation mode start
   :
   G69 ; Coordinate system rotation mode cancel
   G50 ; Scaling mode cancel

2. When the system is in cutter compensation, specify the commands in the following order (Fig.6.11(e)) :
   (cutter compensation cancel)
   G51 ; Scaling mode start
   G68 ; Coordinate system rotation start
   :
   G41 ; Cutter compensation mode start
   :

```
G92 X0 Y0 ;
G51 X300.0 Y150.0 P500 ;
G68 X200.0 Y100.0 R45.0 ;
G01 X400.0 Y100.0 ;
Y100.0 ;
X-200.0 ;
Y-100.0 ;
X200.0 ;
```

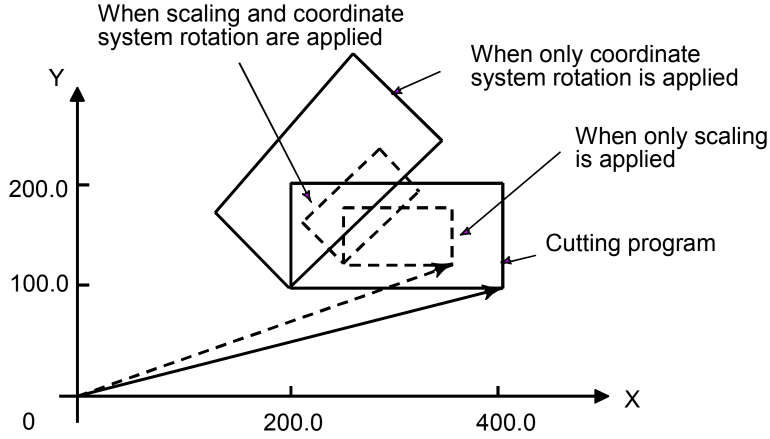


Fig. 6.11 (e) Scaling and coordinate system rotation in cutter compensation mode

- Repetitive commands for coordinate system rotation

It is possible to store one program as a subprogram and recall subprogram by changing the angle.

Sample program for when the RIN bit (bit 0 of parameter 5400) is set to 1.
The specified angular displancement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
G92 X0 Y0 G69 G17;
G01 F200 H01 ;
M98 P2100 ;
M98 P072200 ;
G00 G90 X0 Y0 M30 ;

O 2200 G68 X0 Y0 G91 R45.0 ;
G90 M98 P2100 ;
M99 ;

O 2100 G90 G01 G42 X0 Y-10.0 ;
X4.142 ;
X7.071 Y-7.071 ;
G40 ;
M99 ;
```

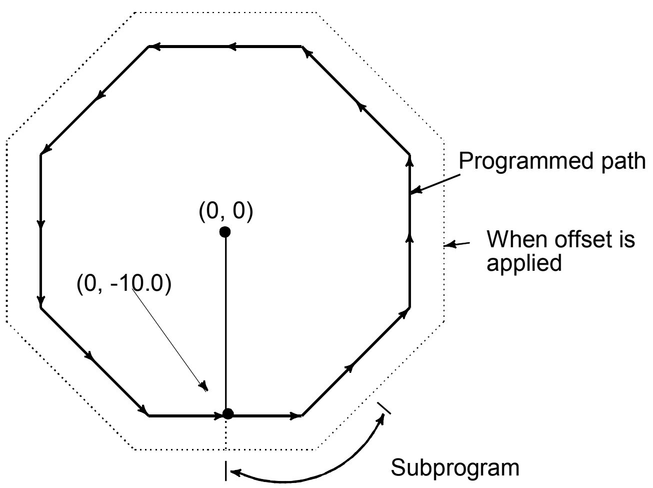


Fig. 6.11 (f) Coordinate system rotation command

6.12 ACTIVE OFFSET VALUE CHANGE FUNCTION BASED ON MANUAL FEED

Overview

When rough machining/semifinish machining is to be performed using a single tool, you may make a fine adjustment of a tool length compensation value or cutter compensation value. Moreover, at setup time, you may want to make a fine adjustment of a workpiece origin offset once set. With this function, a travel distance moved on an axis by manual feed is automatically added to the workpiece coordinate system or the currently valid offset number among the specified offset values (tool length compensation value/cutter compensation value/workpiece origin offset) to make a offset value change.

Explanation

- Active offset value change mode

The active offset value change mode is set using the active offset value change mode signal. In this mode, a travel distance moved on an axis by manual feed is automatically added to the workpiece coordinate system or the currently valid offset number among the specified offset values (tool length compensation value/cutter compensation value/workpiece origin offset). The types of manual feed usable to make an offset value change in this mode are manual handle feed, incremental feed, and jog feed.

> ⚠ **CAUTION**
> 1 When a movement is being made on an axis for which an offset value is to be changed, do not set the active offset value change mode.
> 2 In the active offset value change mode, do not reset the relative coordinate to 0 or preset the relative coordinate to a specified value.

- Specifying an offset value to be changed

The active offset selection signal is used to specify one of three types of offset values: tool length compensation value, cutter compensation value, and workpiece origin offset. In the active offset value change mode, an offset value selected is indicated by blinking display in the state display area on the screen as follows:

Offset value selected	State display
Tool length compensation value	LEN
Cutter compensation value	RAD
Workpiece origin offset	WZR

> ⚠ **CAUTION**
> When a movement is being made on an axis for which an offset value is to be changed in the active offset value change mode, do not change the specification of the offset value to be changed.

- Changing a tool length compensation value

The tool length compensation value with the offset number corresponding to an H code specified in automatic operation is changed. If there is no currently valid tool length compensation value as in a case where no H code is specified after a cycle start, no tool length compensation value change is made even when a movement is made on an axis by manual feed.
With a movement on a linear axis, a tool length compensation value change can be made. With a movement on a rotation axis, no tool length compensation value change can be made. While a tool length compensation value is being changed, a movement by manual feed can be made on one axis only.

> **Example**
> - Specified H code: H10
> - Value set with offset number 10: 54.700 mm
> - Travel distance on the Z-axis by manual feed: -2.583 mm
>
> In this example, the value of offset number 10 becomes:
> 54.700 + (-2.583) = 52.117 mm

> ⚠ **CAUTION**
> A tool length compensation value can be changed by a movement on any linear axis. When an offset value change for an axis is undesirable, interlock the axis.

> **NOTE**
> A changed tool length compensation value is handled according to bit 6 (EVO) of parameter No. 5001 and bit 6 (AON) of parameter No. 5041.

- Changing a cutter compensation value

The cutter compensation value with the offset number corresponding to a D code specified in automatic operation is changed. If there is no currently valid cutter compensation value as in a case where no D code is specified after a cycle start, no cutter compensation value change is made even when a movement is made on an axis by manual feed.
With a movement on a linear axis, a cutter compensation value change can be made. With a movement on a rotation axis, no cutter compensation value change can be made. While a cutter

compensation value is being changed, a movement by manual feed can be made on one axis only.
When operation is stopped in the cutter compensation mode to make a cutter compensation value change, a movable travel distance on one axis is added, regardless of the direction of the compensation vector at stop time.

Example
- Specified D code: H15
- Value set with offset number 15: 6.500mm
- Travel distance on the X-axis by manual feed: 2.379mm
- Travel distance on the Y-axis by manual feed: -0.572mm

In this example, the value of offset number 15 becomes:
6.500+2.379+(-0.572)= 8.307mm

⚠ **CAUTION**
A cutter compensation value can be changed by a movement on any linear axis. When an offset value change for an axis is undesirable, interlock the axis.

NOTE
A changed cutter compensation value is handled according to bit 4 (EVR) of parameter No. 5001.

- Changing a workpiece origin offset value

The workpiece origin offset of the workpiece coordinate system corresponding to a G code from G54 to G59 or from G54.1 P1 to P48 (300) specified during automatic operation is changed on an axis-by-axis basis. A valid workpiece coordinate system exists at all times. So, when a movement is made on an axis by manual feed, the workpiece origin offset of the workpiece coordinate system is changed without fail. This change can be made by a movement on an arbitrary axis, which may be a linear axis or a rotation axis. While a workpiece origin offset change is being made, movements can be made on multiple axes by manual feed.

Example

- Specified workpiece coordinate system : G56
- Workpiece origin offset of G56 (X axis) : 50.000
- Workpiece origin offset of G56 (Y axis) : -60.000
- Workpiece origin offset of G56 (Z axis) : 5.000
- Workpiece origin offset of G56 (A axis) : 5.000
- Workpiece origin offset of G56 (B axis) : 15.000
- Travel distance on the X axis by manual feed : -10.000mm
- Travel distance on the Y axis by manual feed : -5.000mm
- Travel distance on the Z axis by manual feed : 10.000mm
- Travel distance on the A axis by manual feed : 8.000mm
- Travel distance on the B axis by manual feed : -2.000mm

In this example, the workpiece origin offsets of G56 are as follows:

- X axis : 50.000+(-10.000) = 40.000
- Y axis : -60.000+(-5.000) = -65.000
- Z axis : 5.000+10.000 = 15.000
- A axis : 5.000+8.000 = 13.000
- B axis : 15.000+(-2.000) = 13.000

- Operation depending on each tool offset memory

Offset value change operation varies according to tool offset memory A/B/C as follows:

Tool offset memory	Changed offset value
A	No distinction is made between a tool length compensation value and cutter compensation value. The value specified with the offset number corresponding to the currently valid H code or D code is changed.
B	No distinction is made between a tool length compensation value and cutter compensation value. The value specified with the offset number corresponding to the currently valid H code or D code is changed. Depending on the setting of bit 4 (ASG) of parameter No. 5000, a geometry compensation value or wear compensation value is changed.
C	The tool length compensation value and cutter compensation value specified with the offset numbers corresponding to the currently valid H code and D code are changed. Depending on the setting of bit 4 (ASG) of parameter No. 5000, a geometry compensation value or wear compensation value is changed.

- Presetting the relative position indication

By setting bit 5 (APL) of parameter No. 3115 to 1, the relative position indication (counter) can be automatically preset to 0 when the active offset value change mode is selected. In this case, the changed offset value can be restored to the original value by returning the relative position indication (counter) to 0 by manual feed.

- Emergency stop, servo alarm

If an emergency stop occurs, a servo alarm is issued, or servo excitation is turned off, an offset value change is made also for a travel distance on an axis moved by follow-up in the active offset value change mode.

> **NOTE**
> If a tool length compensation value or cutter compensation value is selected as an offset value to be changed, no offset value change is made for a travel distance on a rotation axis moved by follow-up.

Limitation

- Manual operation that cannot change an active offset value

In a mode other than the manual handle feed mode/incremental feed mode/jog feed mode, no active offset value can be changed.
Moreover, no active offset value can be changed in the manual reference position return mode.
Even in the modes mentioned above, do not change an active offset value in the following operations:

- Manual feed for 5-axis machining
- Manual numerical command
- PMC axis control

- Axis that disables an active offset value from being changed

With a rotation axis, no tool length compensation/cutter compensation value can be changed using this function.

6.13 ROTARY TABLE DYNAMIC FIXTURE OFFSET

The rotary table dynamic fixture offset function saves the operator the trouble of resetting the workpiece coordinate system when the rotary table rotates before cutting is started. With this function the operator simply sets the position of a workpiece placed at a certain position on the rotary table as a reference fixture offset. If the rotary table rotates, the system automati-cally obtains a current fixture offset from the angular displacement of the rotary table and creates a suitable workpiece coordinate system. After the reference fixture offset is set, the workpiece coordinate system is prepared dynamically, wherever the rotary table is located.
The zero point of the work piece coordinate system is obtained by adding the fixture offset to the offset from the work piece reference point.

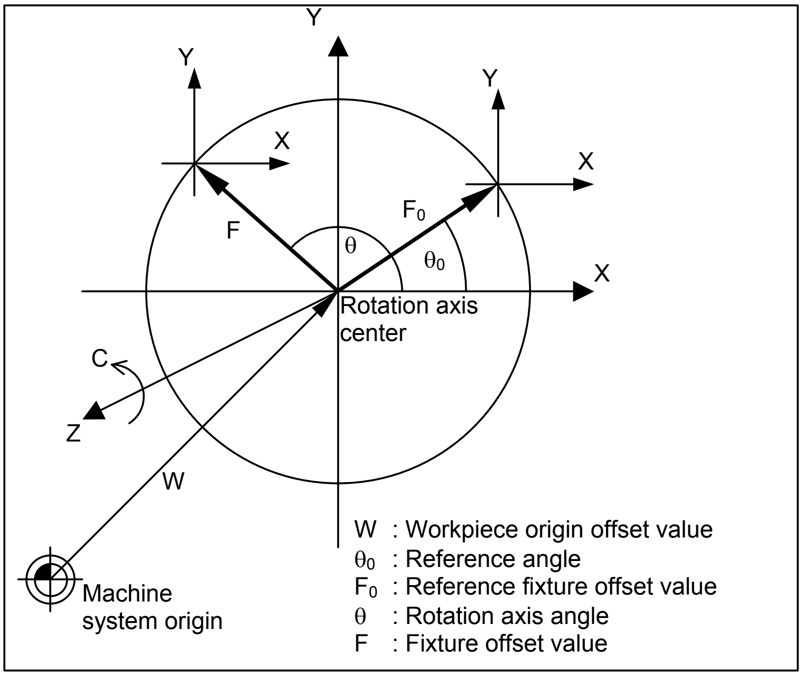


Fig. 6.13 (a) Fixture offset

Format

- Fixture offset command

G54.2 Pn ;
n : Reference fixture offset value number (1 to 8)

- Fixture offset cancel command

G54.2 P0 ;

NOTE

1. In the G54.2 mode, a change made to the setting of parameter or to the reference fixture offset becomes effective when the next G54.2Pn is specified.
2. It depends on the current continuous–state code of the 01 group whether a change in the fixture offset vector causes a movement. If the system is in a mode other than the G00 or G01 mode (G02, G03, etc.), the movement is made temporarily in the G01 mode.
3. When a rotation axis that is related to fixture offset, command is specified in the G54.2 mode, the vector is calculated with the coordinate value of the end of the block and the movment is performed to the command positon on the coordinate system pointed by the vector .
4. In calculation of the fixture offset, the coordinate of the rotation axis on the work piece coordinate system is used. If a tool offset or another offset is applied, the coordinate before the offset is used. If mirror image or scaling is performed, the coordinate before the operation is used.

Explanation

- Fixture offset command

When a command G54.2Pn is specified, a fixture offset value is calculated from the current rotation angle and the data specified with n, and enable the fixture offset value.
If n = 0, the fixture offset value is disabled.

- When a move command is specified for a rotation axis in G54.2 mode

When a command to move the tool about a rotation axis involved with a fixture offset is specified in the G54.2 mode, the coordinates about the rotation axis at the end of the block are used to calculate a vector. The tools moved to the specified position on the work piece coordinate system that is indicated by the vector.

- Operation at reset

Whether to cancel the fixture offset at a reset depends on the settings ofbit 6 (CLR) of parameter No. 3402 and of bit 7 (C23) of parameter No. 3408.
When CLR is set to 0 or CLR and C23 are set to 1, the vector before the reset is saved.
When CLR is set to 1 and C23 is set to 0, the vector is cleared. The machine does not move
by the cleared vector regardless of the setting of bit 0 (FTP) of parameter No. 7570, however.

- Data setting

(1) Setting a group of three parameters which specify one rotation axis and two linear axes constituting the plane of rotation
(Parameter No.7580 to 7588)
In each group, specify the number of the rotation axis as the first parameter and the numbers of the linear axes as the second and third parameters. The rotation in the normal direction about the rotation axis must agree with the rotation from the positive side of the linear axis set as the second parameter to the positive side of the linear axis set as the third parameter.
Example)Suppose that a machine has four axes, X, Y, Z, and C.
The X–, Y–, and Z–axes form a right–handed coordinate system. The C–axis is a rotation axis. When viewed from the positive sideof the Z–axis, a rotation in the normal direction about the C–axis is treated as the counterclockwise rotation around the Z–axis.
For this machine, specify the parameters as follows
First parameter : 4 (C–axis)
Second parameter : 1 (X–axis)
Third parameter : 2 (Y–axis)
Up to three groups of parameters can be set. In calculation of the fixture offset, the data of the rotation axis specified in the first group is calculated first. Then, the data of the second and third groups are calculated.
If a machine has two or more rotation axes and the plane of rotation depends on the rotation about another rotation axis, the plane of rotation is set when the angular displacement about the rotation axis is 0.

(2) Setting the reference angle of the rotation axis and the corresponding reference fixture offset
Set the reference angle of the rotation axis and the fixture offset that corresponds to the reference angle.
Set the data on the fixture offset screen. Eight groups of data items can be specified.

(3) Setting a parameter for enabling or disabling the fixture offset of eachaxis
(bit 0 (FAX) of parameter 7575#0)
For the axis for which the fixture offset is enabled, set the parameter to 1. This need not be specified for a rotation axis.

(4) Setting the type of fixture offset (bit 1 (FTP) of parameter 7570)
Specify whether to cause a movement according to the increment or decrement of the fix-ture offset vector when the vector changes (when G54.2 is specified or when a rotation axis movement occurs in the G54.2 mode).
When 0 is set, the movement is made. (The current position on the workpiece coordinate system does not change. The position on the machine coordinate system changes.)
When 1 is set, the movement is not made. (The current position on the workpiece coordi-nate system changes. The position on the machine coordinate system does not change.)

- Input/output of fixture offset

The data can be programmed and can be input from and output to external equipment, as described below:

(1) Setting the reference fixture offset by G10
G10 L21 Pn P ;
N: Refernece fixture offset number
P: Reference fixture offset or reference angle of each axis
With this command, a reference fixture offset or reference angle can be programmed.
If the command is executed in the G90 mode, the specified value is set directly. If the command is executed in the G91 mode, the sum of the specified value and the previous value is set.

> **NOTE**
> The programmable data input function (G10) function is needed.

(2) Reading/writing based on a custom macro system variable
The following system variable number can be used to read and write a reference fixture offset value or a reference angle. However, it is impossible to write to a system variable area (5500 to 5508) if n = 0.

System variable number = 5500 + 20 * n + m
n: Fixture offset number (1 to 8)
(The current offset is used if n = 0.)
m: Axis number (1 to number of controlled axes)

> **NOTE**
> The custom macro function is needed.

(3) Output to external units
Selecting [PUNCH] on the fixture offset screen enables outputting to external units such as a floppy cassette and memory card via RS-232-C.
Output data is in the G10 format with no program number.

> **NOTE**
> The reader/punch interface and programmable data input (G10) functions are needed.

(4) Input from an external units.
Selecting [READ] on the fixture offset screen, the data can be input from a Floppy Cassette and memory card via RS-232-C.
The data is input in the G10 form with no program number.

> **NOTE**
> The reader/punch interface function and programmable data input functions are needed.

- Calculating a Fixture Offset values

(1) Relationship between the rotation axes and linear axes
First group : 4 (B-axis) , 3 (Z-axis) , 1 (X-axis)
Second group : 5 (C-axis) , 1 (X-axis) , 2 (Y-axis)
Third group : 0 , 0 , 0

(2) Reference angle and reference fixture offset
X : F_{0X}
Y : F_{0Y}
Z : F_{0Z}
B : θ_0
C : ϕ_0

If the above data is set up, the method of calculating fixture offset value is as follows:

O : Rotary table center
W : Workpiece origin offset value
F_0 : Fixture offset value when B=θ_0, C=ϕ_0
F_A : Fixture offset value (F_{AX},F_{AY},F_{AZ}) when B=0, C=0
F : Fixture offset value (F_X,F_Y,F_Z) when B=θ, C=ϕ

Then, the following expression is used for fixture offset calculation.

$$\begin{bmatrix} F_{AX} \\ F_{AY} \\ F_{AZ} \end{bmatrix} = \begin{bmatrix} \cos(-\theta_0) & 0 & \sin(-\theta_0) \\ 0 & 1 & 0 \\ -\sin(-\theta_0) & 0 & \cos(-\theta_0) \end{bmatrix} \begin{bmatrix} \cos(-\phi_0) & -\sin(-\phi_0) & 0 \\ \sin(-\phi_0) & \cos(-\phi_0) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} F_{0X} \\ F_{0Y} \\ F_{0Z} \end{bmatrix}$$

$$\begin{bmatrix} F_{X} \\ F_{Y} \\ F_{Z} \end{bmatrix} = \begin{bmatrix} \cos(\phi) & -\sin(\phi) & 0 \\ \sin(\phi) & \cos(\phi) & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{bmatrix} \cos(\theta) & 0 & \sin(\theta) \\ 0 & 1 & 0 \\ -\sin(\theta) & 0 & \cos(\theta) \end{bmatrix} \begin{bmatrix} F_{AX} \\ F_{AY} \\ F_{AZ} \end{bmatrix}$$

· If manual interventin is made on the rotation axis

When the automatic operation is stopped by the SBK stop or similar in the G54.2 mode, and a manual movement is made about the rotation axis, the vector of the fixture offset does not change. When a rotation axis command is specified in automatic operation or in MDI operation or When G54.2 is specified, the vector is calculated.

When manual intervention is performed with parameter ABS (No. 7570#1) =0 and in the manual absolute switch is set on and then a rotation axis command is specified in the incremental (G91) mode, the vector is calculated using the coordinates which do not reflect the amount of manual intervention.

Example)

N1 G90 G00 C10.0 ;
N2 G54.2 P1 ;

After executing the program, perform manual intervation with the manual absolute swith is set to on. Then, movement of +20.0 about the C-axis.

Afer restart

N3 G91 C30.0 ;

is specified, the coordinate value of C-axis is 60.0 in the workpiece coordinate system.

In the fixture offset calculation, however, the coordinate value of C-axis is considered as 40.0.

When N3 is executed with ABS(No.7570#1) is set to ‘1’. programmed coordninate value of C-axis is used for calculation directly as 40.0(10.0+30.0).

Limitation

- Command for suppressing fixture offset calculation

If the following commands are specified for the rotation axis in the G54.2 mode, the fixture offset vector is not calculated:

Command related to the machine coordinate system: G53
Command specifying a change of the work piece coordinate system: G54 to G59, G54.1, G92, and G52
Command specifying a return to the reference position: G27, G28, G29, G30, G30.1

- Rotation axis used for fixture offset

The rotation axis used for polar coordinate interpolation (G12.1) cannot be set as the rotation axis for the fixture offset.

- Rotation axis roll over

When using the rotary axis roll over function, always specify 360 degrees for the amount of travel per revolution of the rotation axis.

- Functions that cannot be specified

In the G54.2 mode, the functions listed below cannot be specified.

Program restart function
Coordinate system rotation function
Figure copy function

Example

Parameter

Parameter 7580=4 (C–axis)
Parameter 7581=1 (X–axis)
Parameter 7582=2 (Y–axis)
Parameter 7583～7588=0
7575#0(X)=1 (The offset is valid for the X–axis.)
7575#0(Y)=1 (The offset is valid for the Y–axis.)
7570#0=0 (When bit 0 of parameter 7570 is set to 1, the values in square brackets ([ ]) are calculated.)

Data of fixture offset 1 (n = 1)

C= 180.0 (reference angle)
X= −10.0
Y= 0.0

When these parameters and data are set, the machine operates as shown below :

Table 6.13 Example of fixture offset

Program \ Coordinates	Position on the workpiece coordinate system (ABSOLUTE)			Position on the machine coordinate system (MACHINE)			Fixture offset		
	X	Y	C	X	Y	C	X	Y	C
N1 G90 G00 X0 Y0 C90. ;	0.0	0.0	90.0	0.0	0.0	90.0	0.0	0.0	0.0
N2 G54.2 P1 ;	0.0 [0.0	0.0 -10.0	90.0 90.0]	0.0 [0.0	10.0 0.0	90.0 90.0]	0.0 [0.0	10.0 10.0	0.0 0.0]
N3 G01 X10. Y2. F100. ;	10.0	2.0	90.0	10.0	12.0	90.0	0.0	10.0	0.0
N4 G02 X2. Y10. R10. ;	2.0	10.0	90.0	2.0	20.0	90.0	0.0	10.0	0.0
N5 G01 X0 Y0 ;	0.0	0.0	90.0	0.0	10.0	90.0	0.0	10.0	0.0
…									

The values enclosed in brackets ([ ]) apply when bit 0 (FTP) of parameter No. 7570 is set to 1.

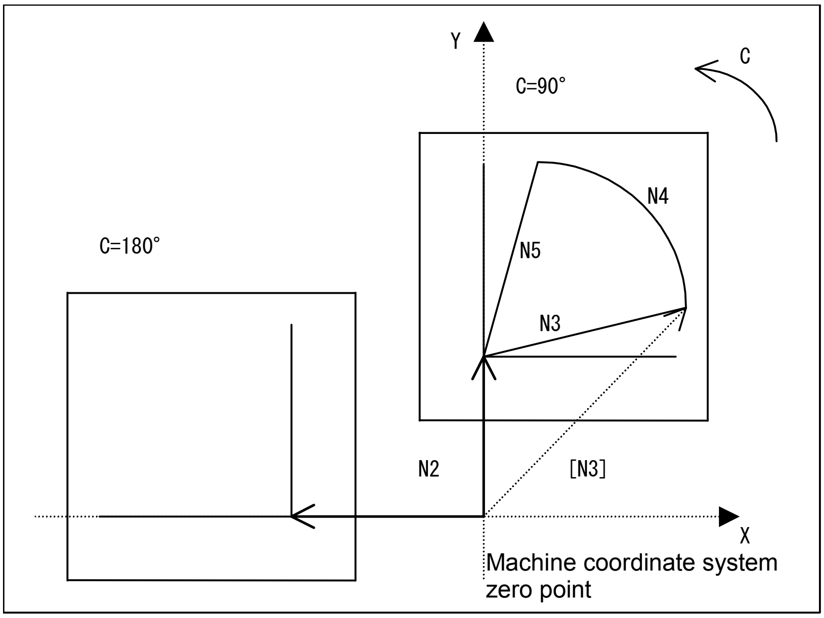


Fig. 6.13 (b) Example of fixture offset

When G54.2 P1 is specified in the N2 block, the fixture offset vector (0, 10.0) is calculated. The vector is handled in the same way as the offset from the workpiece reference point. The current position on the workpiece coordinate system is (0, –10.0). If bit 0 (FTP) of parameter 7570 is set to 0, the tool is moved according to the vector. The resultant position on the workpiece coordinate system is (0, 0), the position before the command is specified.

6.14 NORMAL DIRECTION CONTROL (G40.1, G41.1, G42.1)

Overview

When a tool with a rotation axis (C-axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C-axis is always perpendicular to the tool path (Fig. 6.14 (a)).

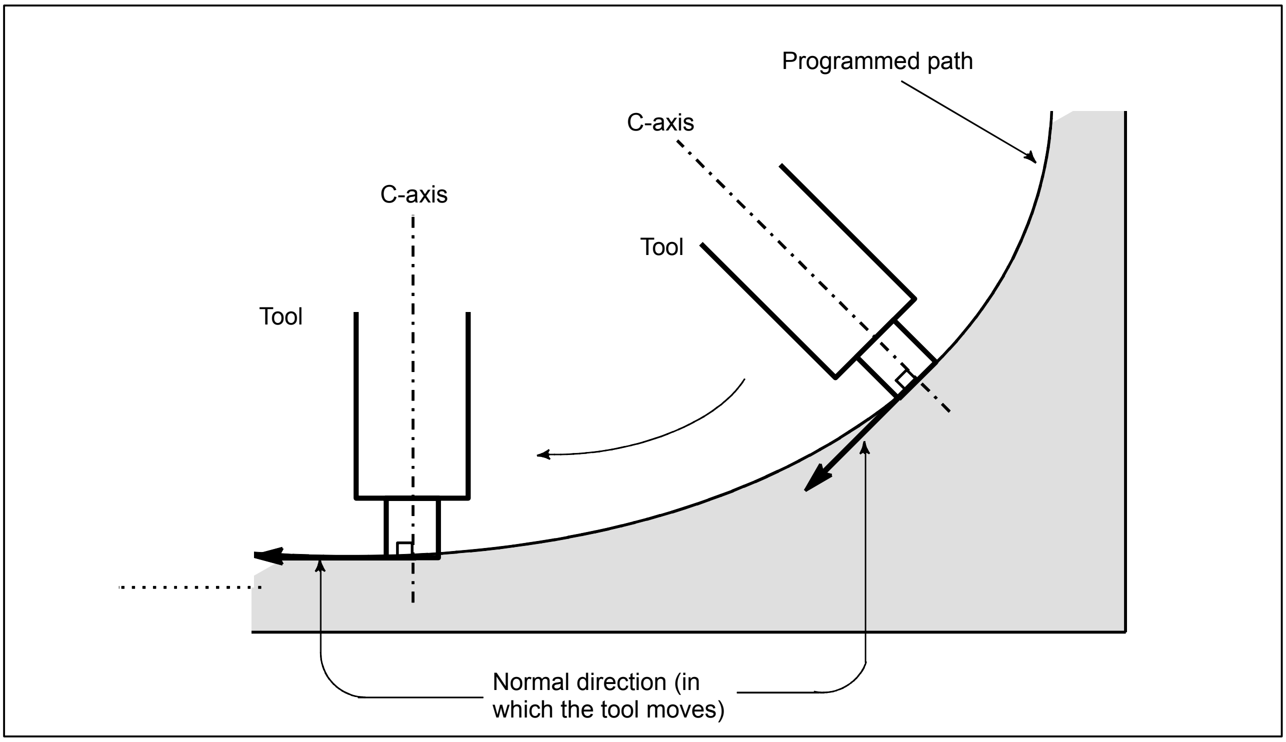


Fig. 6.14 (a) Sample Movement of the tool

Format

G41.1 ; Normal direction control left
G42.1 ; Normal direction control right
G40.1 ; Normal direction control cancel

If the workpiece is to the right of the tool path looking toward the direction in which the tool advances, the normal direction control left (G41.1) function is specified.
After G41.1 or G42.1 is specified, the normal direction control function is enabled (normal direction control mode).
When G40.1 is specified, the normal direction control mode is canceled.

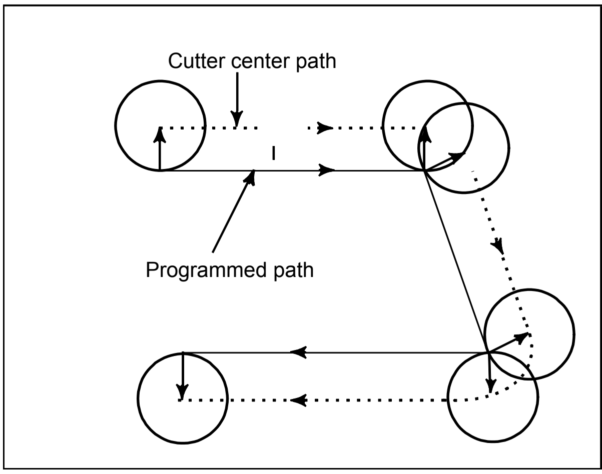


Fig. 6.14 (b) Normal direction control left (G41.1)

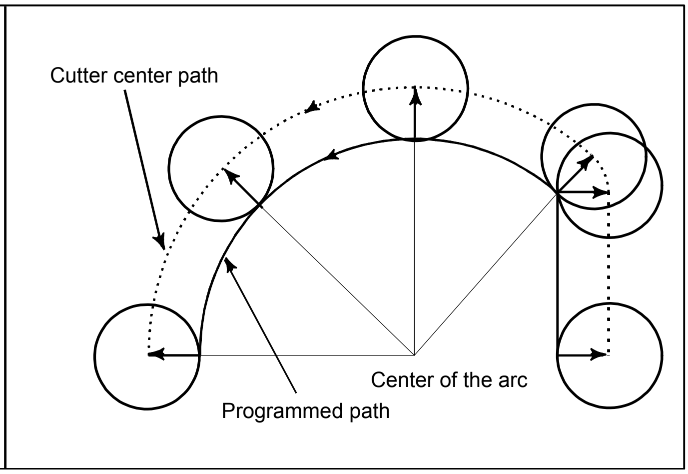


Fig. 6.14 (c) Normal direction control right (G42.1)

Explanation

- Angle of the C axis

When viewed from the center of rotation around the C-axis, the angular displacement about the C-axis is determined as shown in Fig. 6.14 (d). The positive side of the X-axis is assumed to be 0 , the positive side of the Y-axis is 90°, the negative side of the X-axis is 1805, and the negative side of the Y-axis is 270°.

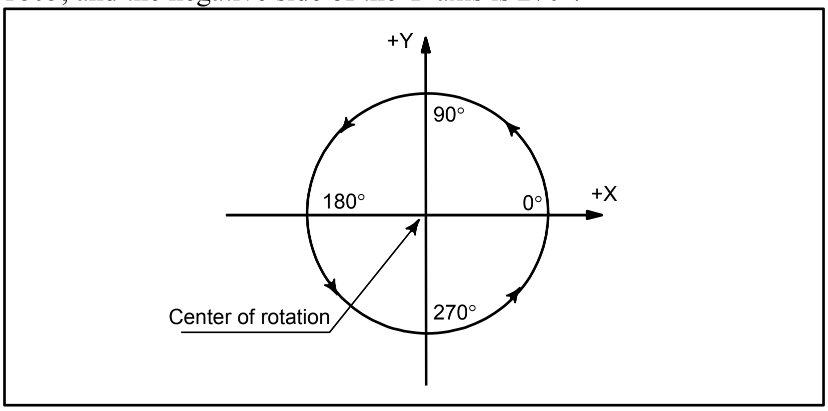


Fig. 6.14 (d) Angle of the C axis

- Normal direction control of the C axis

When the cancel mode is switched to the normal direction control mode, the C-axis becomes perpendicular to the tool path at the beginning of the block containing G41.1 or G42.1.

In the interface between blocks in the normal direction control mode, a command to move the tool is automatically inserted so that the C-axis becomes perpendicular to the tool path at the beginning of each block. The tool is first oriented so that the C-axis becomes perpendicular to the tool path specified by the move command, then it is moved along the X- and Y axes.

In the cutter compensation mode, the tool is oriented so that the C-axis becomes perpendicular to the tool path created after compensation.

In single-block operation, the tool is not stopped between a command for rotation of the tool and a command for movement along the X- and

Y-axes. A single-block stop always occurs after the tool is moved along the X- and Y-axes.

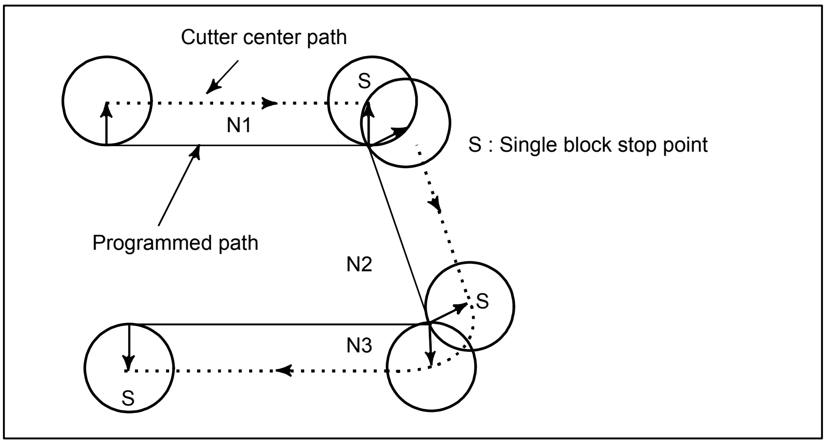


Fig. 6.14 (e) Point at which a single-block stop occurs in the normal direction control mode

Before circular interpolation is started, the C-axis is rotated so that the C-axis becomes normal to the arc at the start point. During circular interpolation, the tool is controlled so that the C-axis is always perpendicular to the tool path determined by circular interpolation.

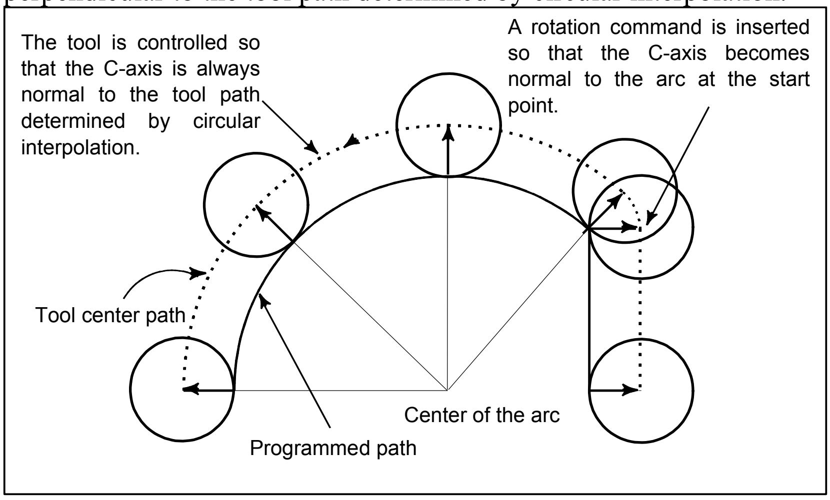


Fig. 6.14 (f) Normal direction control of the circular interpolation

> **NOTE**
> During normal direction control, the C axis always rotates through an angle less than 180 deg. I.e., it rotates in whichever direction provides the shorter route.

- C axis feedrate

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X-and Y-axes in rapid traverse (G00) mode, the rapid traverse feedrate is applied.

The federate of the C axis during circular interpolation is defined by the following formula.

$$F \times \frac{\text{Amount of movement of the C axis (deg)}}{\text{Length of arc (mm or inch)}} \ [\text{deg/min}]$$

F : Federate (mm/min or inch/min) specified by the corresponding block of the arc

Amount of movement of the C axis :

The difference in angles at the beginning and the end of the block.

> **NOTE**
> If the federate of the C axis exceeds the maximum cutting speed of the C axis specified to parameter No. 1430, the federate of each of the other axes is clamped to keep the federate of the C axis below the maximum cutting speed of the C axis.

- Normal direction control axis

A C-axis to which normal-direction control is applied can be assigned to any axis with parameter No. 5480.

- Angle for which figure insertion is ignored

When the rotation angle to be inserted, calculated by normal-direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal-direction control is applied. This ignored rotation angle is added to the next rotation angle to be inserted, the total angle being subject to the same check at the next block.

If an angle of 360 degrees or more is specified, the corresponding rotation block is not inserted.

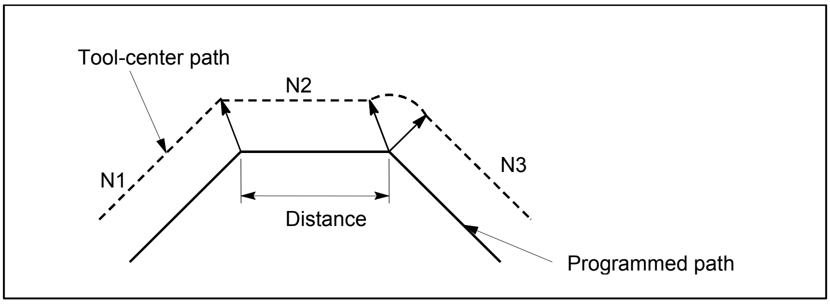
If an angle of 180 degrees or more is specified in a block other than that for circular interpolation with a C-axis rotation angle of 180 degrees or more, the corresponding rotation block is not inserted.

- Movement for which arc insertion is ignored

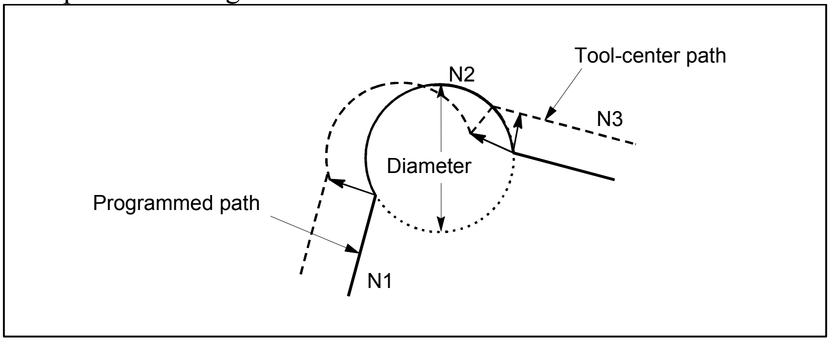
Specify the maximum distance for which machining is performed with the same normal direction as that of the preceding block.

- Linear movement

  When distance N2, shown below, is smaller than the set value, machining for block N2 is performed using the same direction as that for block N1.



- Circular movement
  When the diameter of block N2, shown below, is smaller than the set value, machining for block N2 is performed using the same normal direction as that for block N1. The orientation of the axis to which normal-direction control is applied, relative to the normal direction of block N2, does not change as machining proceeds along the arc.



NOTE
1 Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.
2 Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
3 The helical cutting option is required to use this function. Helical cutting cannot be specified in the normal direction control mode.
4 Normal direction control cannot be performed by the G53 move command.
5 The C-axis must be a rotation axis.

7 MEMORY OPERATION USING Series 15 PROGRAM FORMAT

Overview

Memory operation of the program registered in Series 15 program format is possible by setting the setting parameter FCV (No. 0001#1) to 1.

Explanation

Data formats for cutter compensation, subprogram call, and canned cycles are different between the this CNC and Series 15. The Series 15 program formats can be processed for memory operation.
Other data formats must comply with the Series 30*i*. When a value out of the specified range for the Series 30*i* is registered, an alarm occurs. Functions not available in the Series 30*i* cannot be registered or used for memory operation.

- Address for the cutter compensation offset number

Offset numbers are specified by address D in the Series 15.
When an offset number is specified by address D, the modal value specified by address H is replaced with the offset number specified by address D.

- Subprogram call

If a subprogram number of more than four digits is specified, the four low-order digits are regarded as the subprogram number.
If no repeat count is specified, 1 is assumed.

Table 7 (a) Subprogram call program format

CNC	Program format
Series 15	**M98 P○○○○ L○○○○ ;** P : Subprogram number L : Repetition count (1 to 9999)
Series 30	**M98 P○○○ □□□□ ;** Repetition count (1 to 9999) Subprogram number

- Address for the canned cycle repetition count for drilling

The Series 15 and this CNC use different addresses for the canned cycle repetition count for drilling as listed in Table 7 (b).

Table 7 (b) Address for the canned cycle repetition count for drilling

CNC	Address
Series 15	L
Series 30	K

8 AXIS CONTROL FUNCTIONS

8.1 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis.Positioning is performed by the main motor only. The submotor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

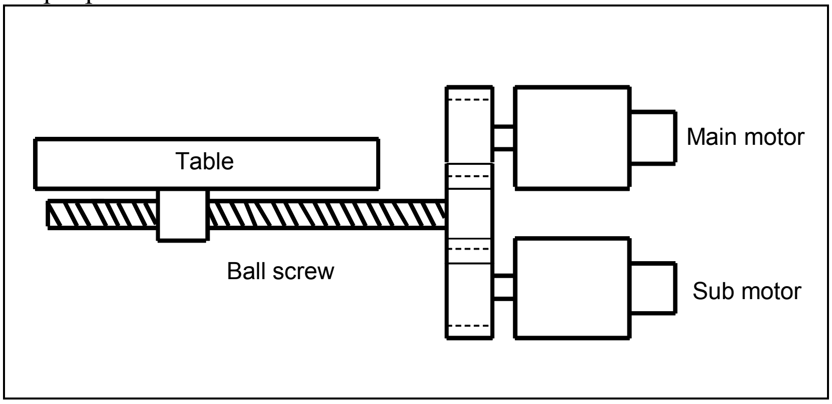


Fig. 8.1 (a) Example of operation

In general, the NC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes.
For details, refer to the relevant manual published by the machine tool builder.

8.2 CHOPPING FUNCTION

Overview

When contour grinding is performed, the chopping function can be used to grind the side face of a workpiece. By means of this function, while the grinding axis (the axis with the grinding wheel) is being moved vertically, a contour program can be executed to initiate movement along other axes.
In addition, a servo delay compensation function is supported for chopping operations. When the grinding axis is moved vertically at high speed, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The servo delay compensation function compensates for any displacement by increasing the feedrate. Thus, grinding can be performed almost up to the specified position.
There are two types of chopping functions: that specified by programming, and that activated by signal input. For details of the chopping function activated by signal input, refer to the manual provided by the machine tool builder.

Format

G81.1 Z_ Q_ R_ F_ ;

Z :Upper dead point
(For an axis other than the Z-axis, specify the axis address.)

Q :Distance between the upper dead point and lower dead point
(Specify the distance as an incremental value, relative to the upper dead point.)

R :Distance from the upper dead point to point R
(Specify the distance as an incremental value, relative to the upper dead point.)

F :Feedrate during chopping

G80 ; Cancels chopping

Explanation

- Chopping activated by signal input

Before chopping can be started, the chopping axis, reference position, upper dead point, lower dead point, and chopping feedrate must be set using the parameter screen (or the chopping screen).

Chopping is started once chopping start signal CHPST has been set to 1. This signal is ignored, however, during chopping axis movement.

When chopping hold signal *CHLD is set to 0 during chopping, the tool immediately moves to point R. Again setting the chopping hold signal to 1 restarts chopping.

Chopping can also be stopped by setting chopping start signal CHPST to 0, but only when chopping was started by using that signal.

Method of starting chopping	Method of stopping chopping	State
Signal CHPST = 1	Signal CHPST = 0	Stopped
	G80	Stopped
G81.1	Signal CHPST = 0	Not stopped
	G80	Stopped

NOTE

1 Switching to manual mode or suspending automatic operation, by means of feed hold, does not stop chopping.

2 In chopping mode, a chopping axis move command or canned cycle command cannot be specified.

3 If a G81.1 command is specified during chopping started by the signal, chopping is not stopped. If point R, the upper dead point, lower dead point, or chopping feedrate has been modified by using the G81.1 command, chopping is continued, but using the modified data.

4 The use of chopping start signal CHPST to start chopping is not enabled immediately after power-on; it is not enabled until the completion of manual reference position return.

- Chopping feedrate (feedrate of movement to point R)

From the start of chopping to point R, the tool moves at the rapid traverse rate (specified by parameter No. 1420).
The override function can be used for either the normal rapid traverse rate or chopping feedrate, one of which can be selected by setting ROV (bit 0 of parameter No. 8360).
When the chopping feedrate is overridden, settings between 110% and 150% are clamped to 100%.

- Chopping feedrate (feedrate of movement from point R)

Between point R, reached after the start of chopping, and the point where the chopping is canceled, the tool moves at the chopping feedrate (specified by parameter No. 8374).
The chopping feedrate is clamped to the maximum chopping feedrate (set with parameter No. 8375) if the specified feedrate is greater than the maximum chopping feedrate.
The feedrate can be overridden by 0% to 150% by applying the chopping feedrate override signal.

- Setting chopping data

Set the following chopping data:

- Chopping axis
  Parameter (No.8370)
- Reference point (point R)
  Parameter (No.8371)
- Upper dead point
  Parameter (No.8372)
- Lower dead point
  Parameter (No.8373)
- Chopping feedrate
  Parameter (No.8374)
- Maximum chopping feedrate
  Parameter (No.8375)

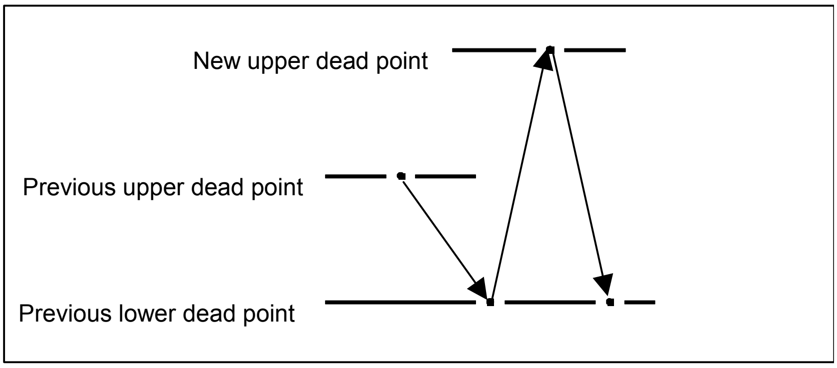
All data items other than the chopping axis and maximum chopping feedrate can be set on the chopping screen.

- Chopping after the upper dead point or lower dead point has been changed

When the upper dead point or lower dead point is changed while chopping is being performed, the tool moves to the position specified by the old data. Then, chopping is continued using the new data.

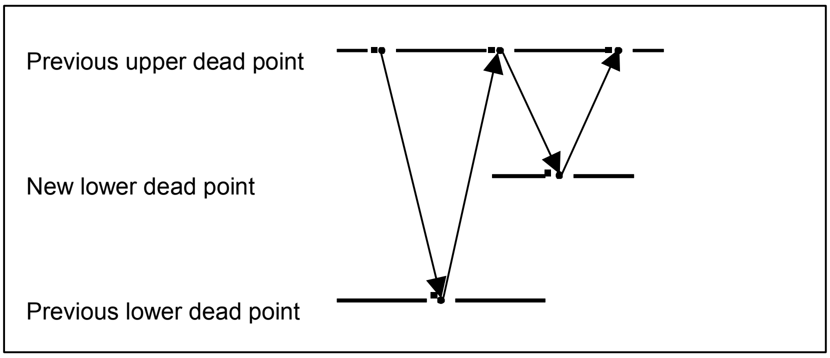
When movement according to the new data starts, the servo delay compensation function stops the servo delay compensation for the old data, and starts the servo delay compensation for the new data.
The following describes the operations performed after the data has been changed.

(1) When the upper dead point is changed during movement from the upper dead point to the lower dead point



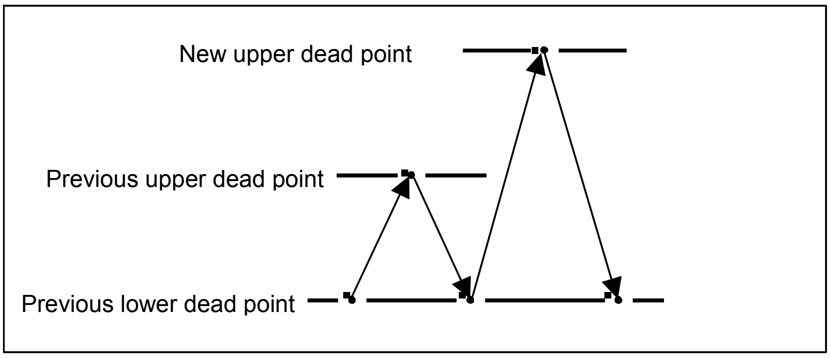
The tool first moves to the lower dead point, then to the new upper dead point.
Once movement to the lower dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(2) When the lower dead point is changed during movement from the upper dead point to the lower dead point



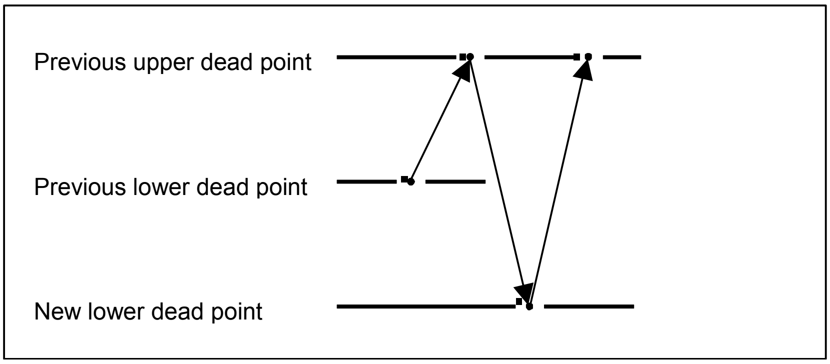
The tool first moves to the previous lower dead point, then to the upper dead point, and finally to the new lower dead point.
Once movement to the upper dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(3) When the upper dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the previous upper dead point, then to the lower dead point, and finally to the new upper dead point.
Once movement to the lower dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

(4) When the lower dead point is changed during movement from the lower dead point to the upper dead point



The tool first moves to the upper dead point, then to the new lower dead point.
Once movement to the upper dead point has been completed, the previous servo delay compensation is set to 0, and servo delay compensation is performed based on the new data.

- Servo delay compensation function

When high-speed chopping is performed with the grinding axis, a servo delay and acceleration/deceleration delay occur. These delays prevent the tool from actually reaching the specified position. The control unit measures the difference between the specified position and the actual tool position, and automatically compensates for the displacement of the tool.
To compensate for this displacement, an amount of travel equal to the distance between the upper and lower dead points, plus an appropriate compensation amount, is specified. When a chopping command is specified, the feedrate is determined so that the chopping count per unit time equals the specified count. When the difference between

the displacement of the tool from the upper dead point and the displacement of the tool from the lower dead point becomes smaller than the setting of parameter No. 8377, after the start of chopping, the control unit performs compensation.

When compensation is applied, the chopping axis moves beyond the specified upper dead point and lower dead point, and the chopping feedrate increases gradually.

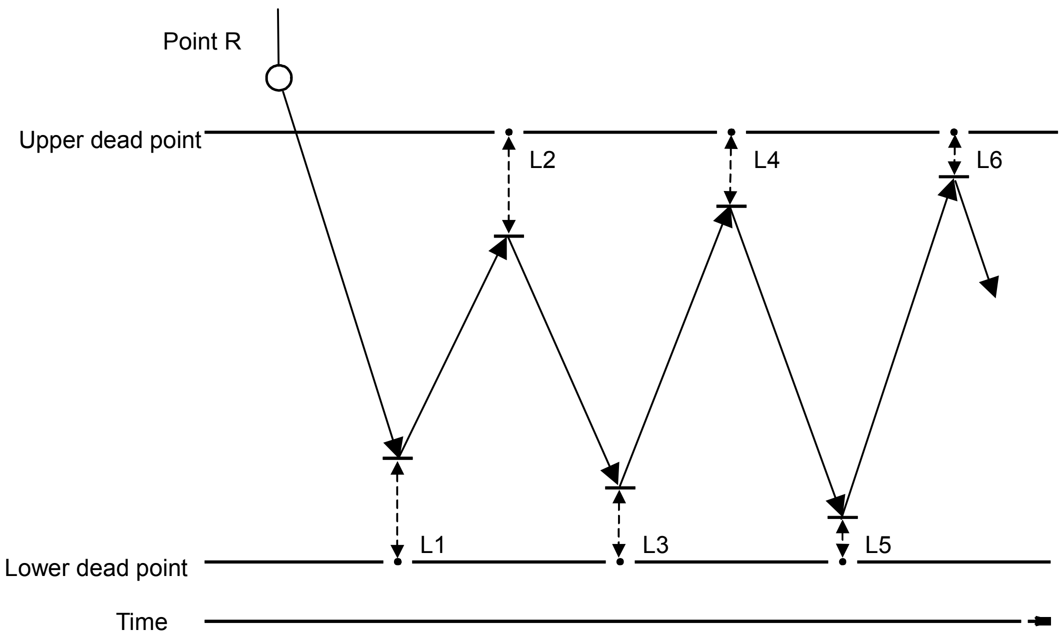
When the difference between the actual machine position and the specified position becomes smaller than the effective area setting (parameter No. 1826), the control unit no longer applies compensation, allowing the tool to continue moving at its current feedrate.

A coefficient for the compensation amount for the displacement generated by the servo delay incurred by chopping and the delay incurred during acceleration/deceleration can be specified in parameter No. 8376.

- If servo delay compensation can cause the chopping speed to exceed the maximum allowable chopping feedrate:

Servo delay compensation during a chopping operation can gradually increase the chopping speed. If the chopping speed is about to exceed the maximum allowable chopping feedrate, it is clamped to the maximum allowable chopping feedrate. In this case, the chopping axis may go beyond the bottom dead point. In servo delay compensation, the distance specified in a movement command is increased by a compensation amount that matches the distance yet to go before the top and bottom dead points are reached, and the chopping speed is also increased, so that the distance yet to go can be compensated for.

If the chopping speed is clamped to the maximum allowable chopping feedrate, a distance specified in the movement command is increased, but the clamped speed remains unchanged. For this reason, the chopping axis can go beyond the bottom dead point.



Displacement between the tool and the upper dead point: L2, L4, L6
Displacement between the tool and the lower dead point: L1, L3, L5
Compensation starts when:
| L3 - L2 | < (parameter No. 8377)
When the following condition is satisfied, compensation is no longer applied, and the tool continues to move at its current feedrate:
| L6 | < effective area setting (parameter No. 1826)

- Acceleration

For the acceleration/declaration along the chopping axis, linear acceleration/deceleration after cutting feed interpolation is effective.

- Mode switching during chopping

If the mode is changed during chopping, chopping does not stop. In manual mode, the chopping axis cannot be moved manually. It can, however, be moved manually by means of the handle interrupt.

- Reset during chopping

When a reset is performed during chopping, the tool immediately moves to point R, after which chopping mode is canceled.
If an emergency stop or servo alarm occurs during chopping, chopping is canceled, and the tool stops immediately.

- Stopping chopping

The following table lists the operations and commands that can be used to stop chopping, the positions at which chopping stops, and the operation performed after chopping stops:

Operation/ command	Stop position	Operation after chopping stops
G80	Point R	Canceled
CHPST : “0”	The tool moves to the lower dead point, then to point R.	Canceled
*CHLD : “0”	Point R	Restart after *CHLD goes "1"
Reset	Point R	Canceled
Emergency stop	The tool stops immediately.	Canceled
Servo alarm	The tool stops immediately.	Canceled
PS alarm	The tool moves to the lower dead point, then to point R.	Canceled
OT alarm	The tool moves from the upper or lower point to point R.	Canceled

- Background editing

When an alarm of background editing or battery alarm is issued, the tool does not stop at point R.

- Single block signal

Even when single block signal SBK is input during chopping, chopping continues.

Limitation

- Workpiece coordinate system

While chopping is being performed, do not change the workpiece coordinate system for the chopping axis.

- PMC axis

When the chopping axis is selected as the PMC axis, chopping is not started.

- Mirror image

While chopping is being performed, never attempt to apply the mirror image
function about the chopping axis.

- Move command during chopping

If a move command is specified for the chopping axis while chopping is being performed, an alarm (PS5050) is issued.

- Program restart

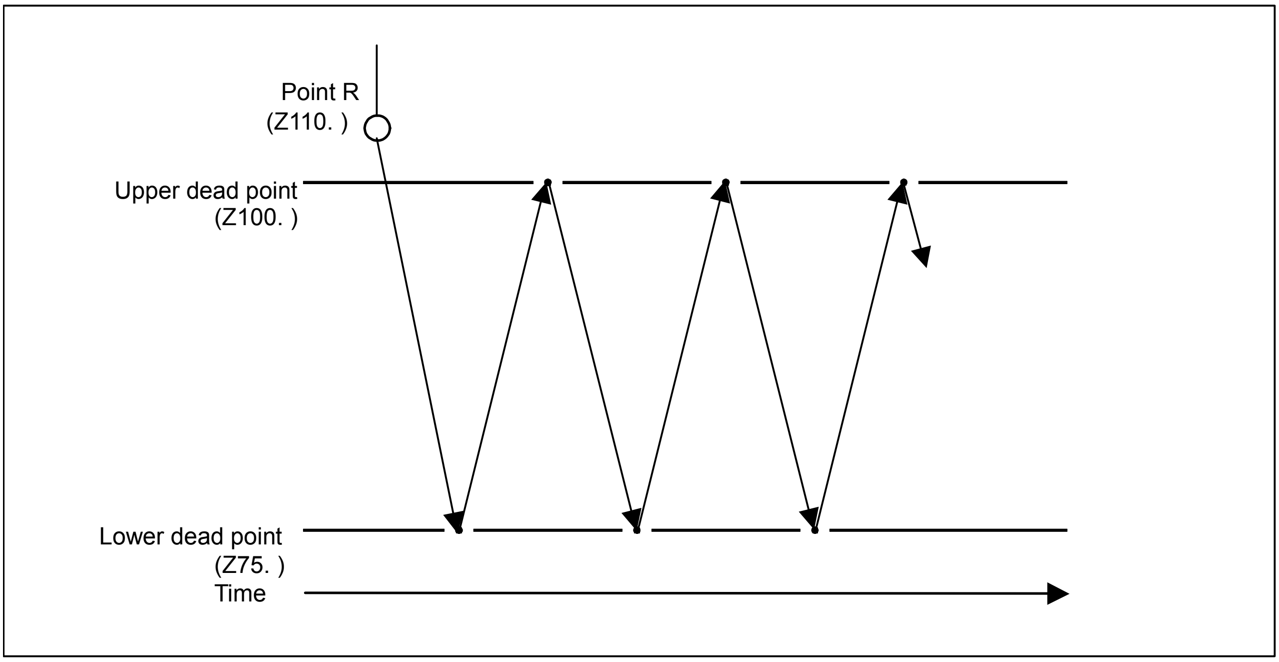
When a program contains G codes for starting chopping (G81.1) and stopping chopping (G80), an attempt to restart that program results in an alarm (PS5050) being output.
When a program that does not include the chopping axis is restarted during chopping, the coordinates and amount of travel set for the chopping axis are not affected after the restart of the program.

Example

Example)

G90 G81.1 Z100. Q-25. R10. F3000 ;

- Perform rapid traverse to position the tool to Z110. (point R).
- Then, perform reciprocating movement along the Z-axis between Z100. (upper dead point) and Z75. (lower dead point) at 3000 mm/min. Chopping override is enabled.



To cancel chopping, specify the following command:

G80 ;

- The tool stops at point R.

OPERATION

1 SETTING AND DISPLAYING DATA

1.1 SCREENS DISPLAYED BY FUNCTION KEY OFFSET SETTING

Press function key OFFSET SETTING to display or set tool compensation values and other data.

This section describes how to display or set the following data:

1. Tool compensation value
2. Tool length measurement
3. Tool length/workpiece origin measurement
4. Rotary table dynamic fixture offset

1.1.1 Setting and Displaying the Tool Compensation Value

Tool offset values, tool length compensation values, and cutter compensation values are specified by D codes or H codes in a program. Compensation values corresponding to D codes or H codes are displayed or set on the screen.

Procedure for setting and displaying the tool compensation value

Procedure

1 Press function key OFFSET SETTING .

For the two-path control, select the path for which tool compensation values are displayed with the tool post selection switch.

2 Press chapter selection soft key [OFFSET] or press OFFSET SETTING several times until the tool compensation screen is displayed.

The screen varies according to the type of tool compensation memory.

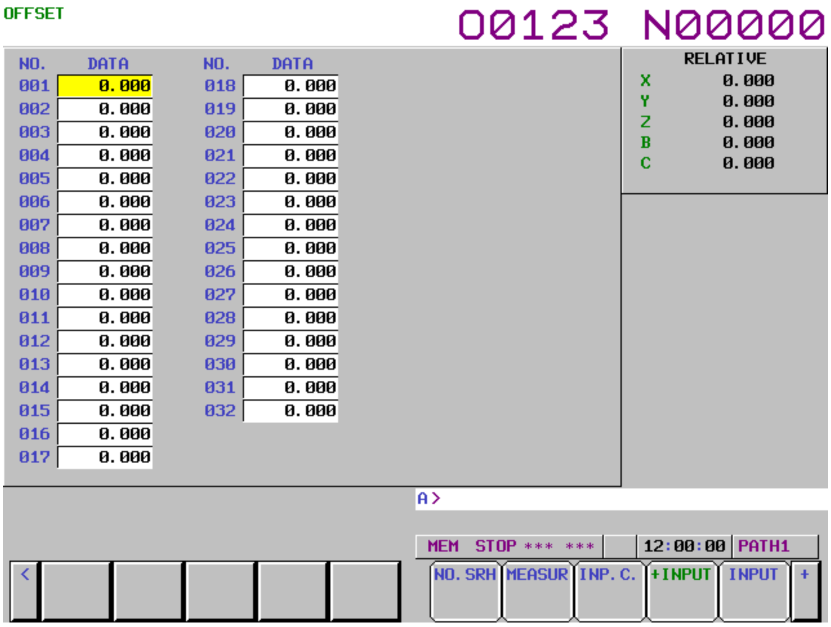


Fig. 1.1.1 (a) Tool compensation memory A

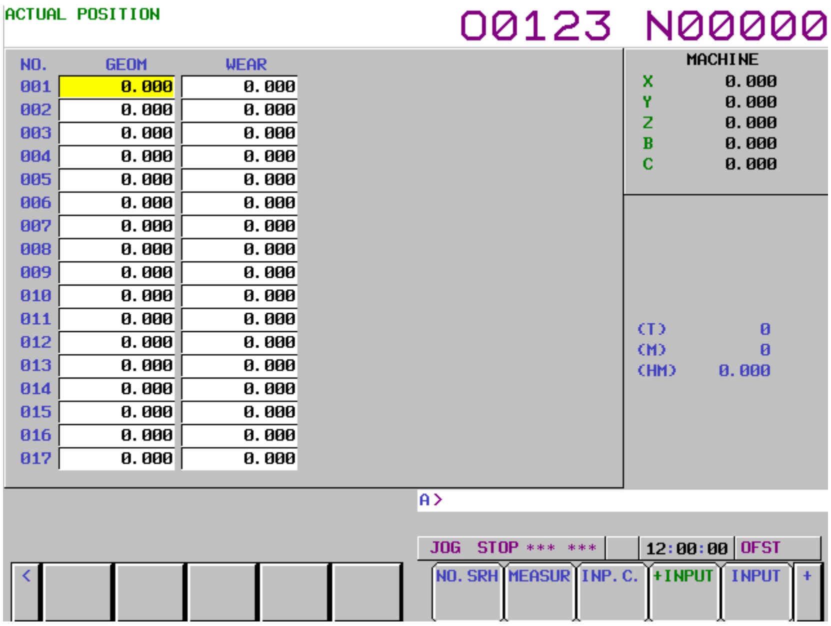


Fig. 1.1.1 (b) Tool compensation memory B

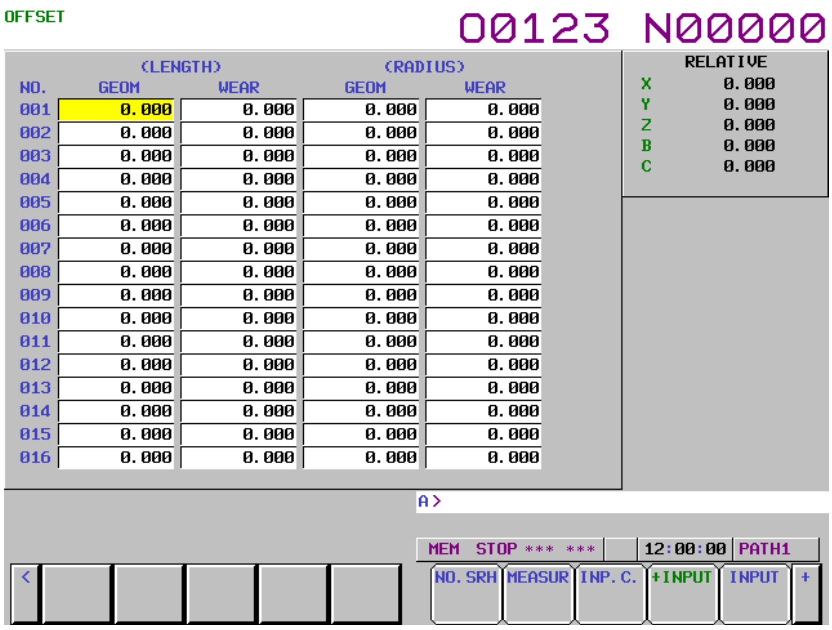


Fig. 1.1.1 (c) Tool compensation memory C

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key [NO.SRH].

4 To set a compensation value, enter a value and press soft key [INPUT].
To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key [+INPUT]. Or, enter a new value and press soft key [INPUT].

Explanation

- Decimal point input

A decimal point can be used when entering a compensation value.

- Other setting method

An external input/output device can be used to input or output a tool offset value. See Chapter III-8 in User's Manual (Common to T/M).
A tool length compensation value can be set by measuring the tool length as described in the next subsection.

- Tool compensation memory

There are tool compensation memories A, B, and C, which are classified as follows:

Tool compensation memory A
  D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated the same.

Tool compensation memory B
  D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated differently.

Tool compensation memory C
  D codes and H codes are treated differently. Tool geometry compensation and tool wear compensation are treated differently.

- Disabling entry of compensation values

The entry of compensation values may be disabled by setting bit 0 (WOF) and bit 1 (GOF) of parameter 3290 (not applied to tool compensation memory A).
And then, the input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.
Consecutive input values are set as follows:

1) When values are input for offset numbers, starting from one for which input is not inhibited to one for which input is inhibited, a warning is issued and values are set only for those offset numbers for which input is not inhibited.
2) When values are input for offset numbers, starting from one for which input is inhibited to one for which input is not inhibited, a warning is issued and no values are set.

1.1.2 Tool Length Measurement

The length of the tool can be measured and registered as the tool length compensation value by moving the reference tool and the tool to be measured until they touch the specified position on the machine. The tool length can be measured along the X-, Y-, or Z-axis.

Procedure for tool length measurement

Procedure

1. Use manual operation to move the reference tool until it touches the specified position on the machine (or workpiece.)
2. Press function key POS several times until the current position display screen with relative coordinates is displayed.

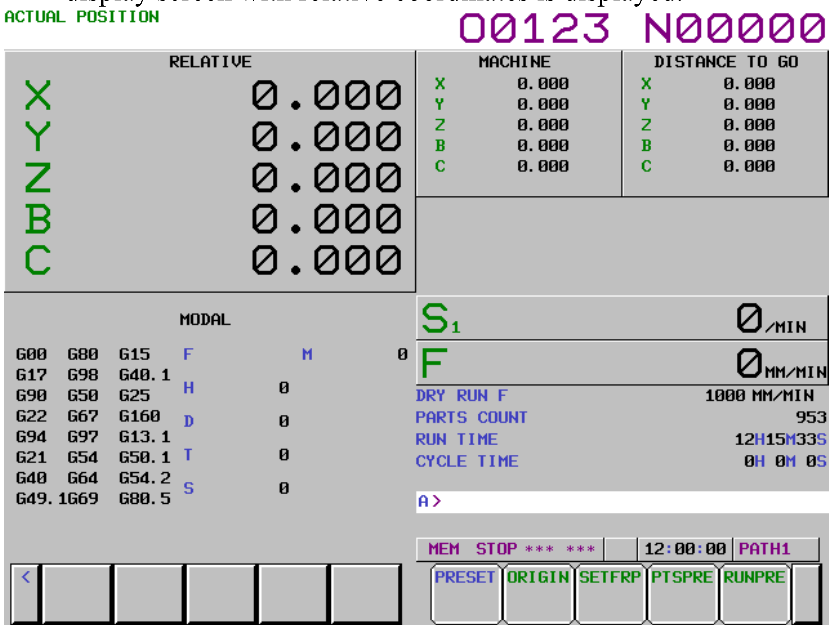
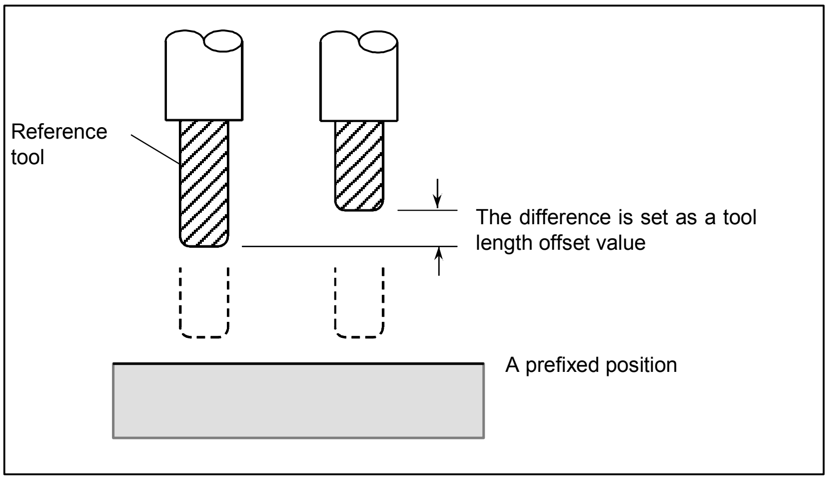


Fig. 1.1.2 (a)

3. Reset the relative coordinate for the Z-axis to 0.
4. Press function key OFFSET SETTING several times until the tool compensation screen is displayed.
5. Use manual operation to move the tool to be measured until it touches the same specified position. The difference between the length of the reference tool and the tool to be measured is displayed in the relative coordinates on the screen.
6. Move the cursor to the compensation number for the target tool (the cursor can be moved in the same way as for setting tool compensation values).
7. Press the address key Z . If either X or Y key is depressed instead of Z key, the X or Y axis relative coordinate value is input as an tool length compensation value.

8 Press the soft key [INP.C.]. The Z axis relative coordinate value is input and displayed as an tool length compensation value.



1.1.3 Tool Length/Workpiece Origin Measurement B

To enable measurement of the tool length, the following functions are supported: automatic measurement of the tool length by using a program command (G37) (automatic tool length measurement, described in Section II-6.2) and measurement of the tool length by manually moving the tool until it touches a reference position, such as the workpiece top surface (tool length measurement, described in Subsection III-1.1.2). In addition to these functions, tool length/workpiece origin measurement B is supported to simplify the tool length measurement procedure, thus facilitating and reducing the time required for machining setup. This function also facilitates the measurement of the workpiece origin offsets.
This function allows the operator to specify T/M code commands or reference position return, by means of a manual numeric command, while the tool length compensation measurement screen is displayed.

Procedure for measuring the tool length compensation value

Procedure

The tool length compensation value can be measured by manually moving the tool until it touches the workpiece or a reference block. For details of this operation, refer to the manual supplied by the machine tool builder.

1 Move the tool to the tool change position by means of manual reference position return, for example.
2 Press mode selection switch HANDLE or JOG.
3 Set the tool offset value measurement mode switch on the machine operator's panel to ON. The tool length compensation measurement screen, shown below, appears and "OFST" blinks in the status display at the bottom of the screen.
  The tool length compensation measurement screen varies slightly depending on whether tool length compensation memory A, B (geometry compensation and wear compensation are treated differently), or C (geometry compensation and wear compensation are treated differently, and cutter compensation and tool length compensation are treated differently) is used.

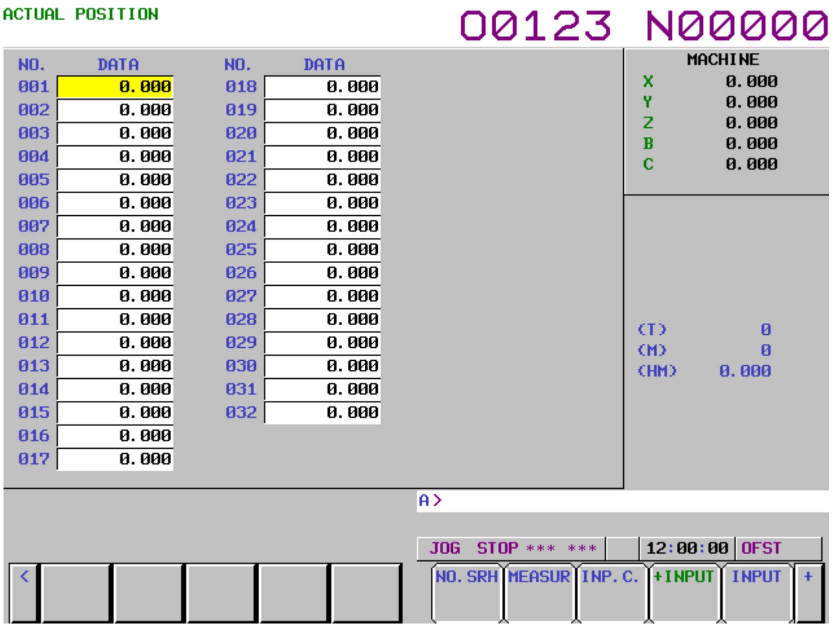


Fig. 1.1.3 (a) Tool length compensation measurement screen for tool compensation memory A

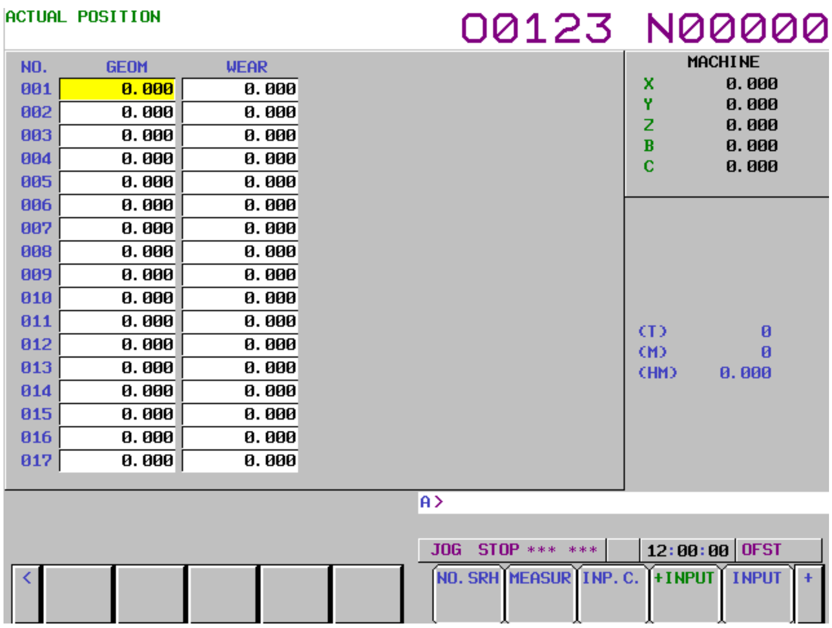


Fig. 1.1.3 (b) Tool length compensation measurement screen for tool compensation memory B

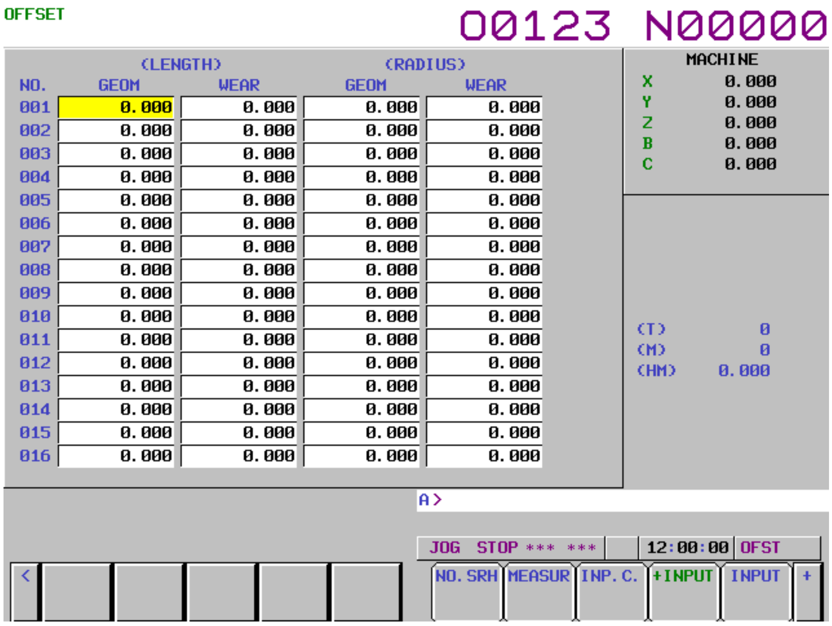


Fig. 1.1.3 (c) Tool length compensation measurement screen for tool compensation memory C

> **NOTE**
>
> Pressing the RESET key resets the displayed T and M addresses to 0. Once MEM or MDI mode has been selected, however, the modal T and M codes are displayed.

4 Use the numeric keys to enter the distance from the base measurement surface to the measurement surface, then press soft key [HM INPUT] to set the distance. For details of the measurement surface and base measurement surface, see Explanations, below.

5 Select the tool for which the tool length compensation value is to be measured.
While "OFST" is blinking at the bottom of the tool length compensation measurement screen, a T code or M code can be specified in manual handle feed or jog feed mode (manual numeric command). First, enter Ttttt (where tttt is a T code number), then press the cycle start button on the machine operator's panel or MDI panel. The Ttttt command is executed, thus selecting the tool to be measured. Then, usually, enter the M06 command to move the tool to the spindle position. Once the tool for which the tool length compensation is to be measured has been selected at the spindle position, position the cursor to the tool offset number with which the tool length compensation for the selected tool is to be stored. The positioning of the cursor to the offset number is usually done by the operator. Some machines, however, automatically position the cursor to an appropriate tool offset number upon the completion of tool selection, if bit 5 (QNI) of parameter No. 5005 is set to 1.

6 Perform manual handle feed or jog feed to move the tool until it touches the measurement surface of the workpiece or reference block.
7 Press soft key [MEASUR]. The tool length compensation is stored in the tool compensation memory. If tool compensation memory B or C is being used, the tool length compensation is set as the tool geometry value, while 0 is set as the tool wear offset. The cursor remains positioned to the set tool offset number. To automatically advance the cursor to the next tool offset number upon the completion of the setting an offset, press soft key [MEASUR+], instead of [MEASUR].
8 Once the tool length compensation has been set, the tool is automatically moved to the tool change position.
9 This completes tool length compensation measurement for a single tool. To measure the tool length compensations of other tools, repeat steps 5 to 8.
10 Once the tool length compensations of all tools have been measured, set the tool offset measurement mode switch on the machine operator's panel to OFF. The "OFST" blinking indication is cleared from the bottom of the screen.

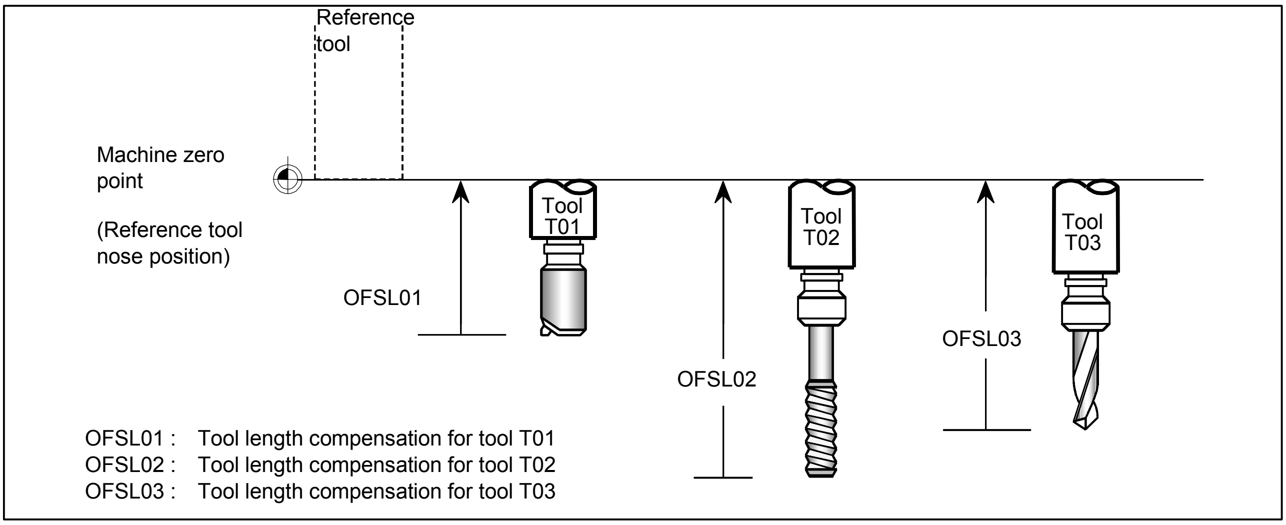
Explanation

- Definition of tool length compensation value

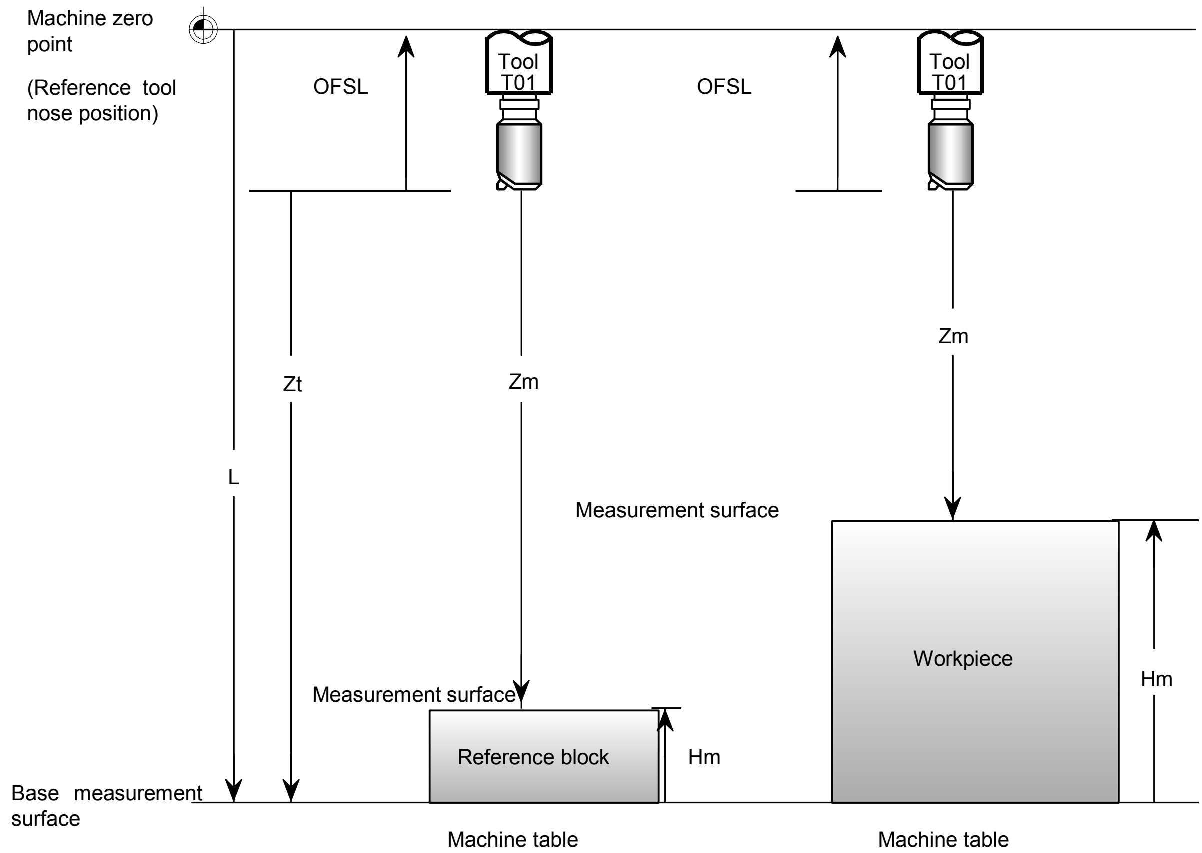
In general, the tool length compensation value can be defined in either of the following two ways. Both methods are based on the same concept: The difference between the tool nose position of the tool and that of a reference tool is used as the tool offset.

(1) Definition 1

The first method involves using the actual tool length as the tool length compensation. In this case, the reference tool is an imaginary tool which has its tool nose at the machine zero point when the machine is positioned to the Z-axis machine zero point. The difference between the tool nose position of the tool to be measured and that of the reference tool, that is, the distance along the Z-axis from the machine zero point to the tool nose when the machine is positioned to the Z-axis machine zero point, is defined as the tool length compensation.



Also, with this function, the tool is manually moved by means of jog feed until its tool nose touches the top surface of the workpiece or reference block. This surface is called the measurement surface. Assume that the top surface of the machines table is set as the measurement surface, although this is actually not allowed because the machine would be damaged. In such a case, distance L from the machine zero point to the machine table top surface is specific to that machine. Set distance L in a parameter (No. 5022). Assume Zt to be the machine coordinate of the tool at the position where it would touch the machine table top surface if that surface were set as the measurement surface. The tool length compensation (OFSL) can then be easily calculated from L and Zt. Because the machine table top surface cannot actually be used as the measurement surface, however, that surface is defined as the base measurement surface and the distance from the base measurement surface to the actual measurement surface, that is, the height of the workpiece or reference block (Hm) must be set. The tool length compensation value (OFSL) can thus be obtained from the formula shown below.

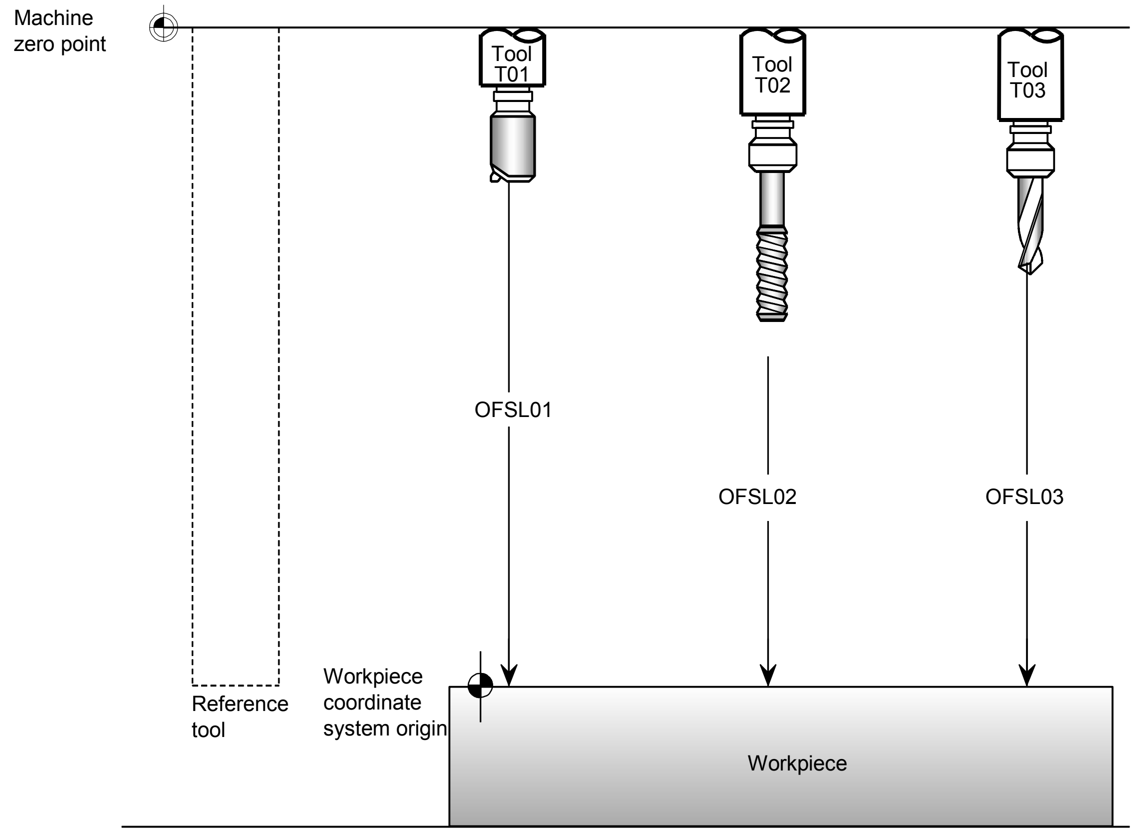


L : Distance from the reference tool nose position to the base measurement surface (machine coordinate of the measurement surface)
Hm : Distance from the base measurement surface to the actual measurement surface
Zm : Distance from the tool nose to be measured to the measurement surface when the tool is positioned to the machine zero point
(Zt : Distance from the tool nose to be measured to the base measurement surface when the tool is positioned to the machine zero point)
OFSL: Tool length compensation value (OFSL = Zm - Hm - L)

Defining the actual tool length as the tool length compensation has the advantage of eliminating the need for remeasuring, even if the workpiece is changed, provided the tool is not worn. Another advantage is that the tool length compensation need not be re-set when multiple workpieces are machined. In this case, assign a workpiece coordinate system to each workpiece, using G54 to G59, and set the workpiece origin offset for each workpiece.

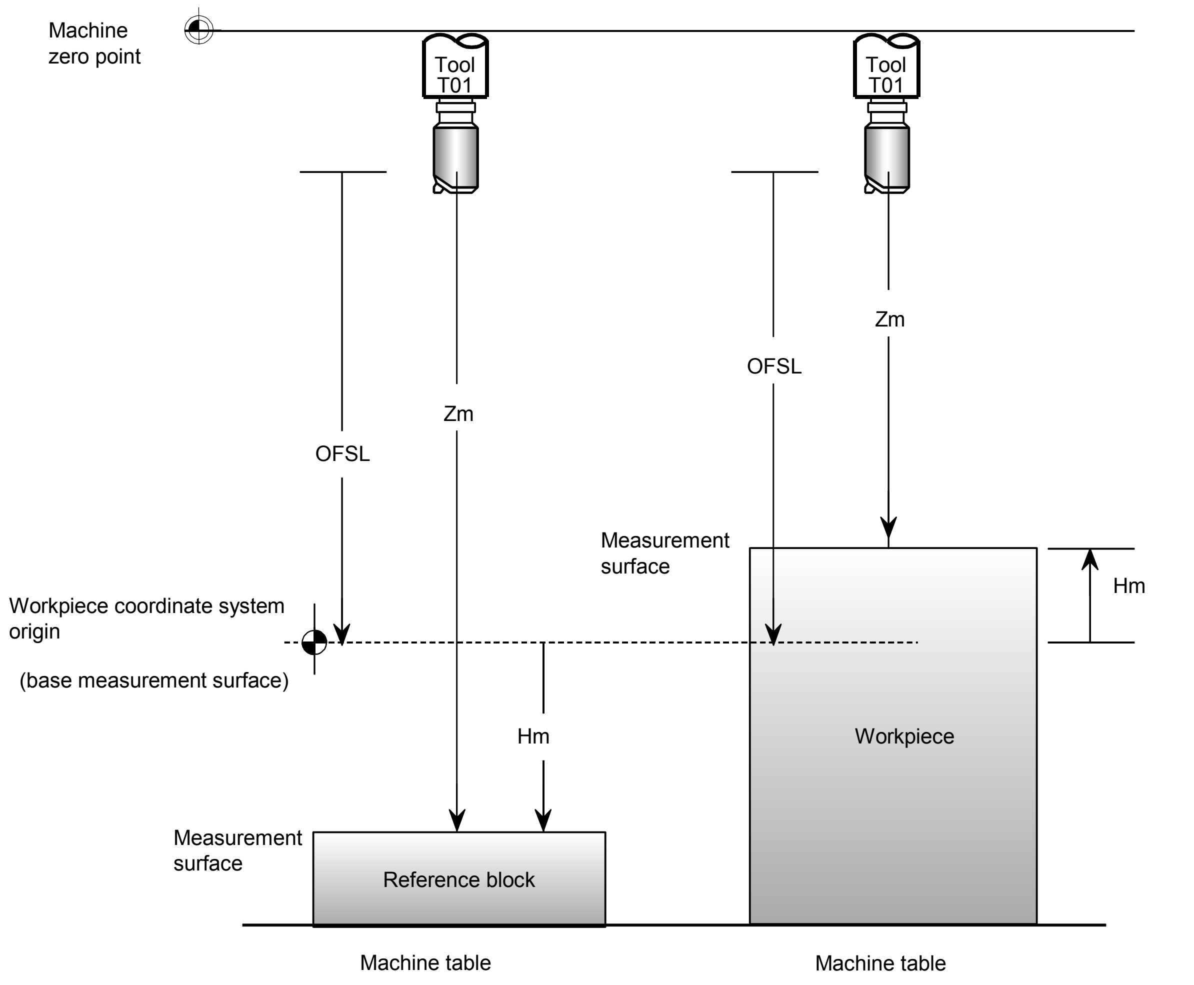
(2) Definition 2
In the second definition method, the tool length compensation is the distance from the tool nose position to the workpiece coordinate system origin when the machine is positioned to the Z-axis zero point. A tool length compensation defined in this way will be equal to the difference between the length of the tool to be measured and that of the reference tool, in the same way as with definition 1. The reference tool for definition 2 is, however, an imaginary tool which has a tool nose at the workpiece

coordinate system origin when the machine is positioned to the Z-axis zero point.



OFSL01 : Tool length offset for tool T01
OFSL02 : Tool length offset for tool T02
OFSL03 : Tool length offset for tool T03

The base measurement surface for this definition is located at the workpiece coordinate system origin. Because the of the reference tool nose is also located at the workpiece coordinate system origin, distance L from the reference tool nose position to the base measurement surface is 0. Set, therefore, 0 in the parameter for distance L (No. 5022). The actual measurement surface is usually the same as the base measurement surface, located at the workpiece coordinate system origin. If, however, the measurement surface is the top surface of the reference block, or if the workpiece coordinate system origin is located on other than the top surface of the workpiece (for example, when the origin is shifted from the workpiece top surface by an amount equal to the cutting allowance), set the distance from the base measurement surface to the actual measurement surface as Hm, such that the tool length compensation (OFSL) can be calculated using the same formula as that used for definition 1.



L : Distance from the reference tool tip position to the base measurement surface (= 0)
Hm : Distance from the base measurement surface to the actual measurement surface
Zm : Distance from the tip of the tool to be measured to the measurement surface when the tool is positioned to the machine zero point
OFSL: Tool length offset (OFSL = Zm - Hm - L)

The reference tool for definition 2 has a tool nose at the workpiece coordinate system origin when the machine is positioned to the Z-axis zero point. Whenever the workpiece is changed, therefore, the tool length compensation must be remeasured. Remeasuring is not, however, necessary if the difference between the workpiece coordinate system origin for a new workpiece and that when the tool length compensation value was measured is set as the new workpiece origin offset (any of G54 to G59). In such a case, the tool length compensation need not be modified, even when the workpiece is changed.

Taking a different point of view, definition 2 can be thought of as setting the workpiece origin offset as the tool length compensation for each tool.

- Measuring the tool length compensation along a specified axis

Because the tool is usually mounted in parallel with the Z-axis, the tool length compensation is measured by moving the tool along the Z-axis. Some machines, however, have their W-axis in parallel with the Z-axis, making it necessary to measure the tool length compensation by moving the tool along the W-axis. Moreover, some machines, when fitted with an attachment, support the mounting of the tool in parallel with an axis other than the Z-axis. For such a machine, the tool length compensation can be measured along a specified axis by setting bit 2 (TMA) of parameter No. 5007 to 1. To measure the tool length compensation along an axis other than the Z-axis, first set distance L from the reference tool nose position to the base measurement surface, for each of the axes along which the tool length compensation may be measured, in parameter No. 5022, in addition to distance L along the Z-axis. Next, set distance Hm from the base measurement surface to the actual measurement surface for the axis along which the tool length compensation is to be measured (see Explanations, below). Finally, move the tool along that axis until it touches the workpiece or reference block, then enter the name of that axis before pressing soft key [MEASUR] or [MEASUR+]. When the tool offset is measured along the W-axis, for example, enter W then press soft key [MEASUR] or [MEASURE+].

- Tool change position

The tool change position must be set beforehand, using bits 1 (TC3) and 0 (TC2) of parameter No. 5007.

Table 1.1.3 (a)

TC3	TC2	Meaning
0	0	The tool change position is the first reference position (G28)
0	1	The tool change position is the second reference position (G30 P2)
1	0	The tool change position is the third reference position (G30 P3)
1	1	The tool change position is the fourth reference position (G30 P4)

Procedure for measuring the workpiece origin offset

In addition to the workpiece origin offset along the tool lengthwise axis, that is, the Z-axis, the workpiece origin offsets along the X- and Y-axes, on a plane perpendicular to the Z-axis, can also be measured easily. The workpiece origin offsets along the X- and Y-axes can be measured regardless of whether the workpiece origin is located on a surface of the workpiece or at the center of a hole to be machined. For details of this measurement, refer to the manual supplied by the machine tool builder.

- Measuring the Z-axis workpiece origin offset

1 Select a tool using an MDI command, then move it to the spindle position (see the explanation of the procedure for measuring the tool length compensation). The tool length compensation for the selected tool must be measured beforehand.

2 Press mode selection switch HANDLE or JOG.

3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The workpiece origin offset screen appears and "WOFS" blinks in the status display at the bottom of the screen.

4 Enter the tool length compensation for the selected tool. Enter the offset using numeric keys then press soft key [TL INPUT].

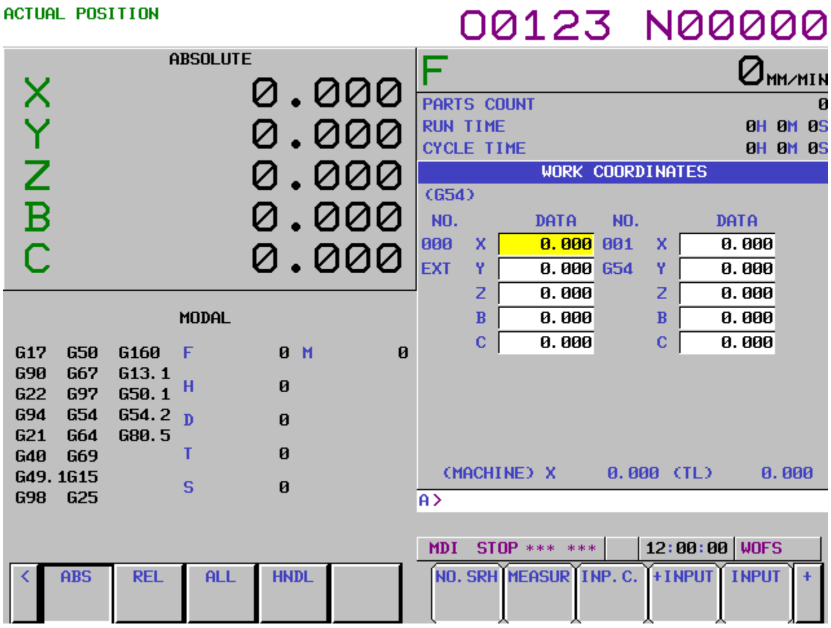


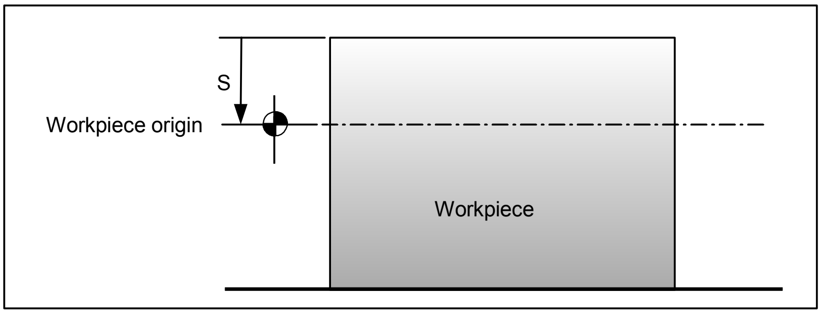
Fig. 1.1.3 (d) Workpiece origin offset setting screen

5 Position the cursor to the workpiece origin offset number to be used to store the offset (any of G54 to G59). No problem will arise even if the cursor is positioned to the offset for other than the Z-axis.

6 Move the tool by means of manual handle feed or jog feed until it touches the top surface of the workpiece.

7 Enter the axis name, Z, press soft key [MEASUR], then press soft key [INPUT]. The Z-axis workpiece origin offset value is set and the cursor is positioned to the set Z-axis workpiece origin offset. There is no need to enter Z provided the parameter has been set so that only the Z-axis workpiece origin offset is to be measured (bit 3 (WMA) of No. 5007 = 0).

To set the workpiece origin on other than the workpiece top surface (for example, when the origin is shifted from the workpiece top surface by an amount equal to the cutting allowance), enter the amount of shift (S in the following figure) using the numeric keys, press soft key [MEASUR], then press soft key [INPUT].



8 To measure any subsequent workpiece origins, retract the tool from the workpiece, then repeat steps 5 to 7.

- Measuring the X-/Y-axis workpiece origin offset based on a reference surface

To set the X- or Y-axis workpiece origin offset on a specified surface of the workpiece, set bit 3 (WMA) of parameter No. 5007 to 1, then follow the same procedure as that for measuring the Z-axis workpiece origin offset. In step 4, however, enter the cutter compensation value for the selected tool, instead of the tool length compensation. After entering the cutter compensation value with the numeric keys, press soft key [TL INPUT].

> ⚠ **CAUTION**
> When entering the cutter compensation value, ensure that its sign is entered correctly.
> - When the measurement surface is located in the positive (+) direction relative to the tool, enter a minus (-) sign.
> - When the measurement surface is located in the negative (-) direction relative to the tool, enter a plus (+) sign.

- Measuring the X-/Y-axis workpiece origin offset based on a reference hole

1 Connect a measurement probe, fitted with a sensor, to the spindle.

2 Press mode selection switch HANDLE or JOG.

3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The workpiece origin offset screen appears and "WOFS" blinks in the status display at the bottom of the screen, indicating that the preparation required prior to measuring the workpiece origin offset has been completed.

4 Position the cursor to the workpiece origin offset number to be used to store the offset (any of G54 to G59). No problem will arise even if the cursor is positioned to the offset for other than the X- or Y-axis.

5 Move the tool by means of manual handle feed or jog feed until the measurement probe touches the circumference of the hole. Do not move the tool along more than one axis at any one time.

6 As soon as the sensor detects contact with the circumference, input a skip signal to the machine, thus stopping the axial movement of manual handle feed or jog feed. Simultaneously, the position at which feed stopped is stored as the first measurement point. The machine coordinates of the stored measurement point are displayed at the bottom right of the screen, as follows:

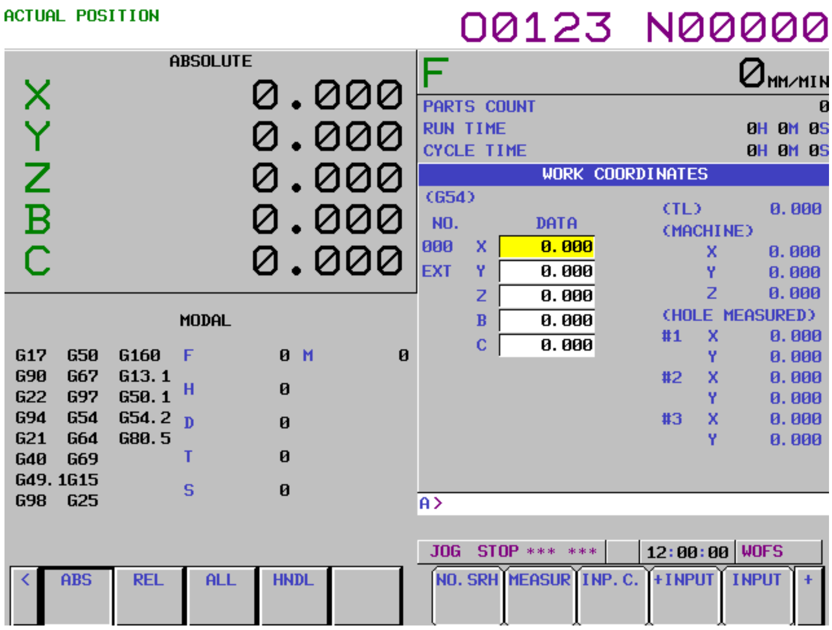


Fig. 1.1.3 (e) Workpiece origin offset setting screen

7 Move the measurement probe to the second measurement point. At this time, the CNC interlocks the machine to prevent the probe from moving in the direction in which it was moved so as to touch the current measurement point. For example, when the probe touched the measurement point after being moved in the +X direction, movement of the probe to the next measurement point is allowed only in the -X direction. Movement in the +X, +Y, or -Y direction is interlocked until the skip signal is set to 0. Once the probe touches the second measurement point, follow the same procedure as that for storing the first measurement point.

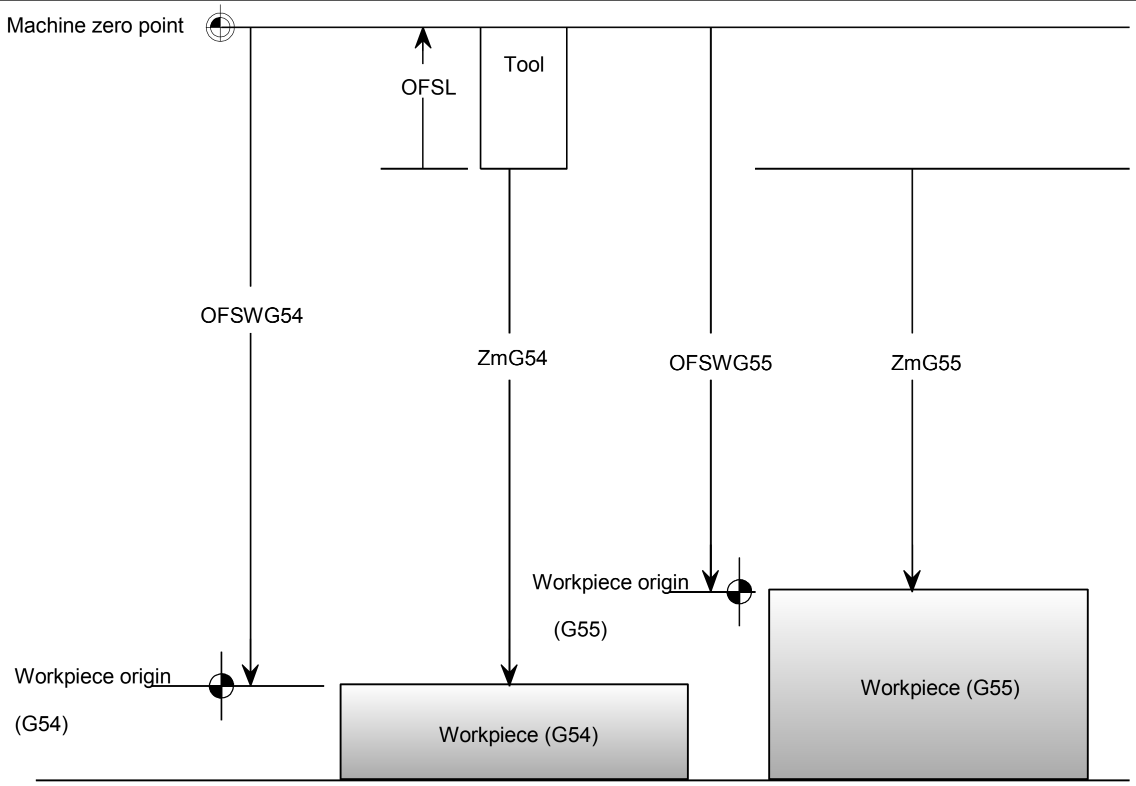
8 Once the probe has touched the third measurement point, press soft key [MEASUR], then [CENTER]. This calculates the center of the hole from the coordinates of the three measured points, then sets the X- and Y-axis workpiece origin offsets. To cancel and restart measurement at any point, press the RESET key. Pressing the RESET key clears the coordinates of all stored measurement points.

Explanation

- Z-axis workpiece origin offset

Definitions 1 and 2, described in "Definition of tool length compensation" in Explanations for measuring the tool length compensation, also apply to the general concept of the Z-axis workpiece origin offset, as follows:

(1) Definition 1
In definition 1, the Z-axis workpiece origin offset is defined as the distance from the machine zero point to the origin of the workpiece coordinate system.

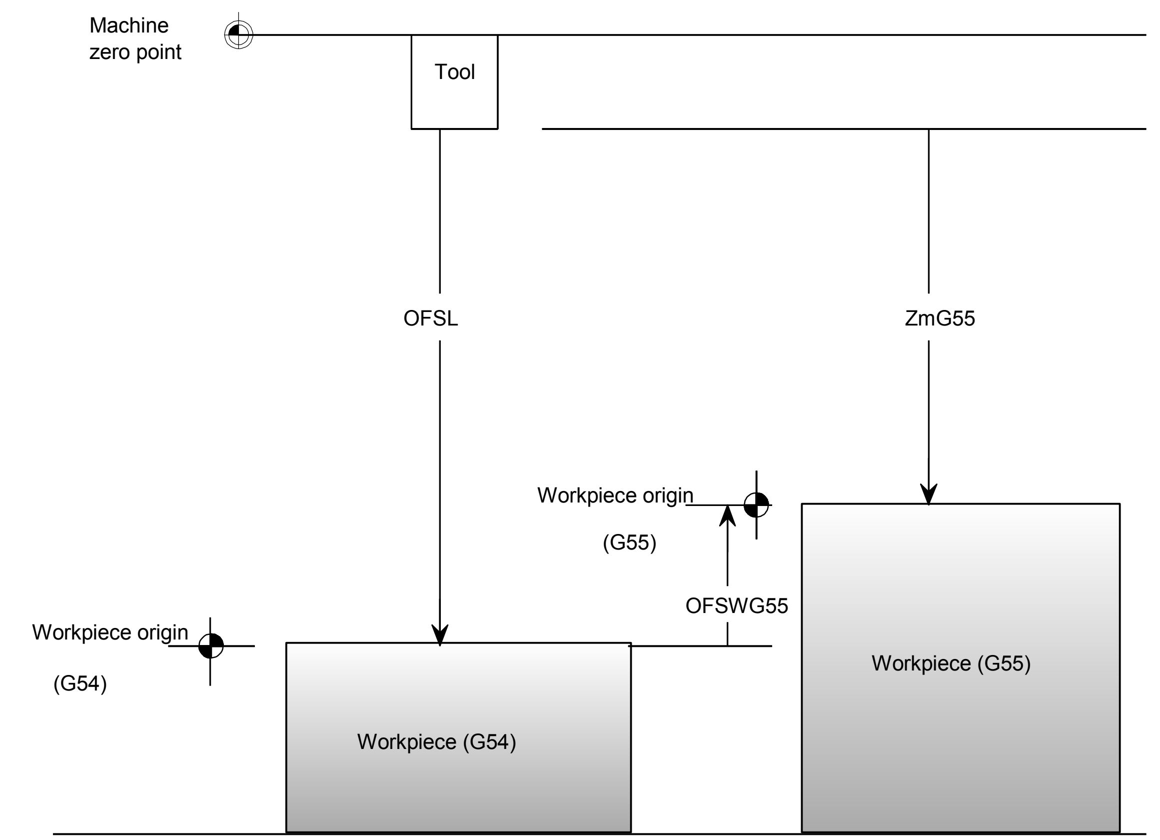


OFSL : Tool length compensation for the tool used to measure the workpiece origin offset
ZmG54 : Amount of movement from the machine zero point to the origin of the G54 workpiece when measured with a tool having a length of OFSL
ZmG55 : Amount of movement from the machine zero point to the origin of the G55 workpiece when measured with a tool having a length of OFSL
OFSWG54 : Workpiece origin offset for the G54 workpiece
OFSWG55 : Workpiece origin offset for the G55 workpiece

As can be seen from the above figure, the Z-axis workpiece origin offset can be calculated from the following formula:
OFSW=Zm-OFSL
where
OFSW : Workpiece origin offset
OFSL : Tool length compensation for the tool used to measure the workpiece origin offset
Zm : Amount of movement from the machine zero point to the workpiece origin when measured with a tool having a length of OFSL

(2) Definition 2

The tool length compensation in definition 2 equals the Z-axis workpiece origin offset, as described above. Usually in this case, therefore, the workpiece origin offset need not be set. If, however, the workpiece is changed after its tool length compensation has been measured, or if multiple workpieces are machined, the workpiece origin coordinates can be set as follows when assigning workpiece coordinate systems to G54 to G59, thus eliminating the need to remeasure the tool length compensation.



OFSL : Tool length compensation measured for the G54 workpiece
ZmG55 : Amount of movement from the machine zero point to the origin of the G55 workpiece when measured with a tool having a length of OFSL
OFSWG55 : Workpiece origin offset for the G55 workpiece (The workpiece origin offset for the G54 workpiece is 0.)

For definition 2, the workpiece origin offset can be calculated using the same formula as that used for definition 1:

OFSW = Zm - OFSL

where

OFSW: Workpiece origin offset

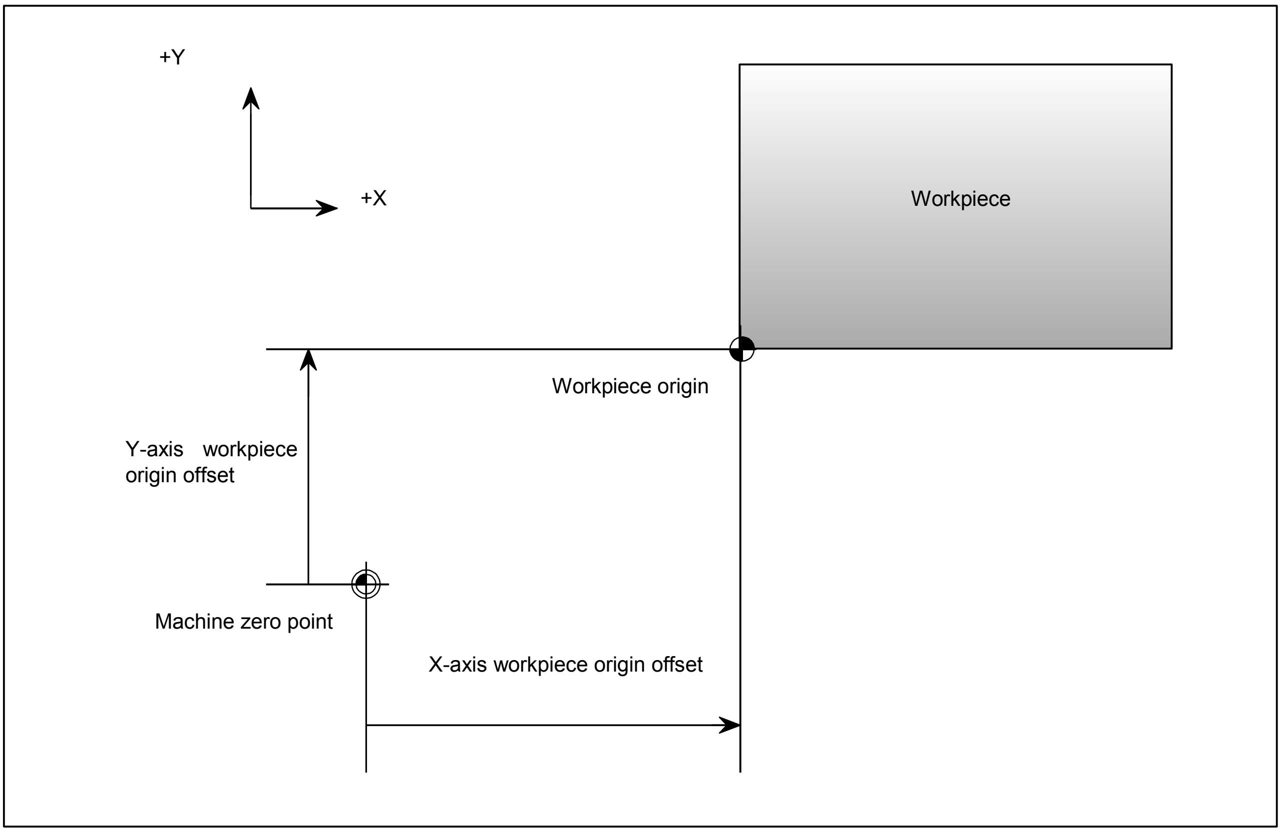
OFSL : Tool length compensation for the tool used to measure the workpiece origin offset

Zm : Amount of movement from the machine zero point to the workpiece origin when measured with a tool having a length of OFSL

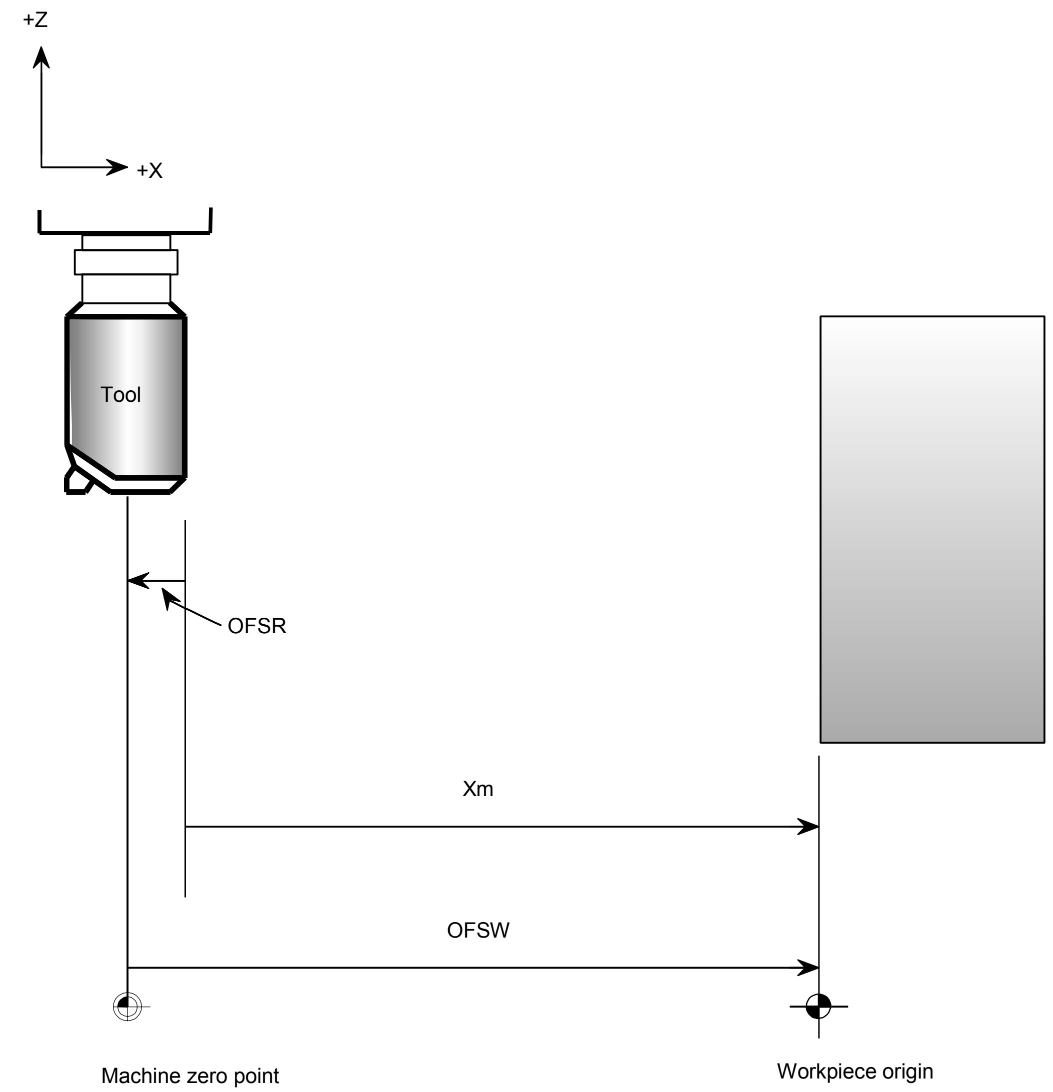
- X-/Y-axis workpiece origin offset

The X- and Y-axis workpiece origin offsets can be measured regardless of whether the workpiece origin is located on a surface of the workpiece or at the center of a hole to be machined.

(1) When the workpiece origin is located on a surface



In the above case, the workpiece origin is located on a side surface of the workpiece. The measurement of the X-/Y-axis workpiece origin offset when the origin is located on a surface of the workpiece is the same as that for the Z-axis workpiece origin offset, but with the following exception: The tool length compensation for the tool used to measure the offset is used to calculate the Z-axis workpiece origin offset, while the cutter compensation value for the tool is used to calculate the X-/Y-axis workpiece origin offset.



OFSR : Cutter compensation value for the tool used to measure the workpiece origin offset
Xm : Amount of movement from the machine zero point to the workpiece origin when measured with a tool having a length of OFSR
OFSW : Workpiece origin offset

As can be seen from the above figure, the workpiece origin offset can be calculated from the following formula:

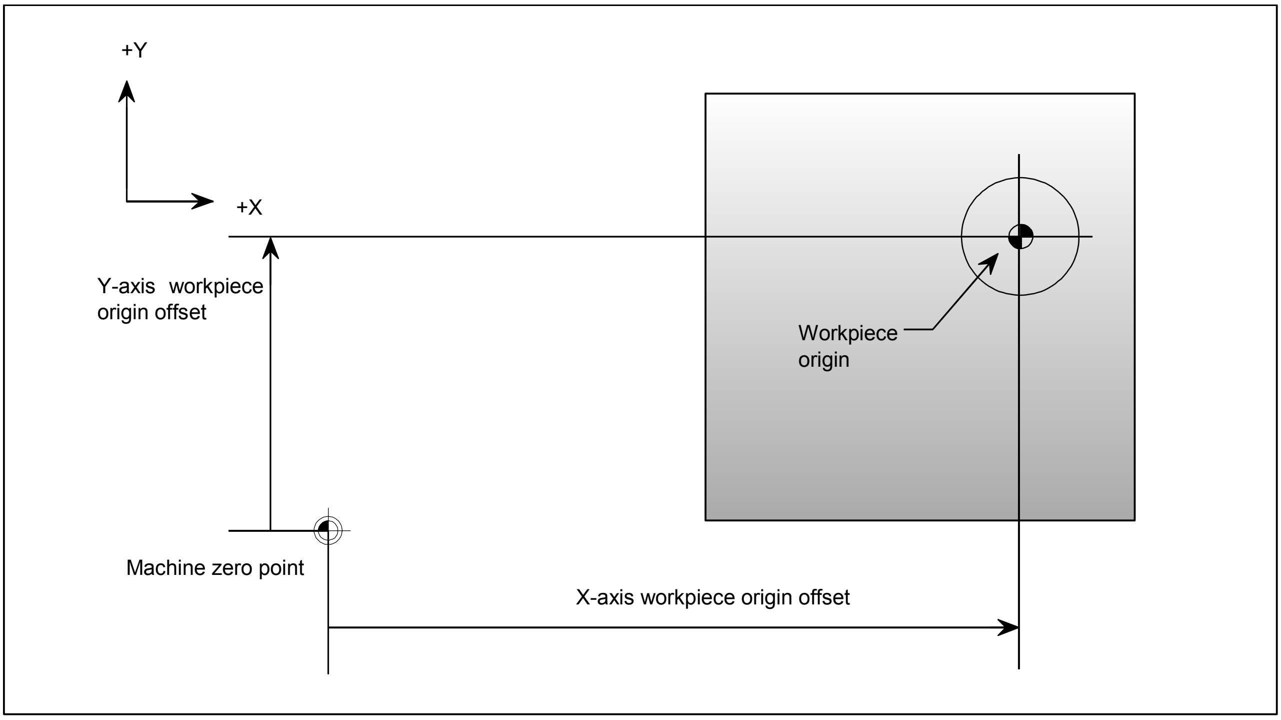
OFSW = Xm - OFSR

Pay particularly careful attention, however, to the sign of the cutter compensation value OFSR:

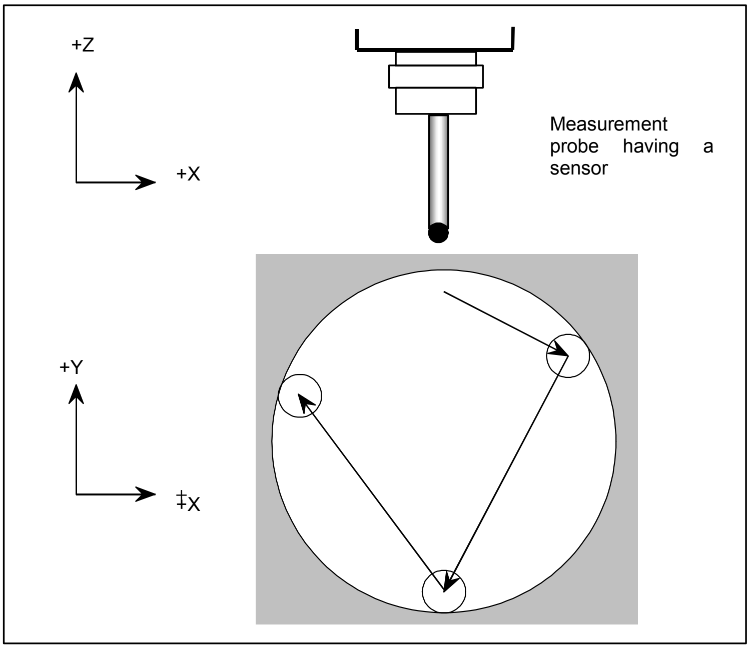
The sign of OFSR is - when the measurement surface is located in the positive (+) direction relative to the tool center.

The sign of OFSR is + when the measurement surface is located in the negative (-) direction relative to the tool center.

(2) When the workpiece origin is located at the center of a hole.



In the above case, the workpiece origin is located at the center of a hole in the workpiece. A measurement probe having a sensor at its tool nose is used to measure the positions of three arbitrary points on the circumference of the hole. The three points prescribe a unique circle, the center of which is set as the X-/Y-axis workpiece origin. Set bit 4 (WMH) of parameter No. 5007 to 1 before starting the measurement.



- Using a skip signal

A measurement probe, fitted with a sensor, can also be used to measure the Z-axis workpiece origin offset or measure the X-/Y-axis workpiece origin offset based on a surface, in the same way as when measuring the X-/Y-axis workpiece origin offset based on a hole. By inputting a skip signal as soon as the probe touches the workpiece surface, feed is automatically stopped. Subsequently, apply the same procedure as that for each measurement.

1.1.4 Setting and Displaying the Rotary Table Dynamic Fixture Offset

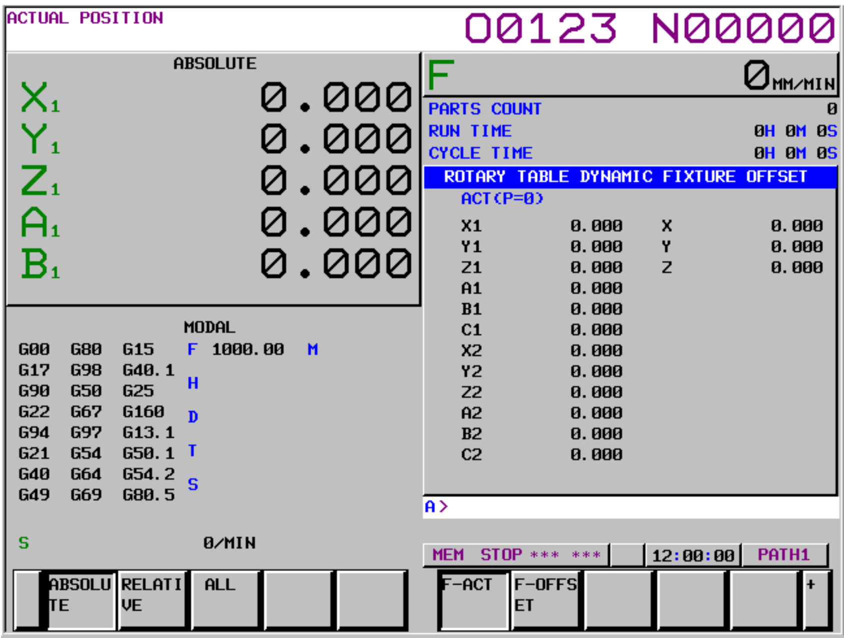
The fixture offset screen is either a fixture offset (ACT) screen for verifying the currently selected fixture offset value or a fixture offset screen for setting and verifying eight fixture offset value sets.

Ative Fixture offset screen

Procedure

1. Press function key OFFSET SETTING .
2. Press the continuous menu key ▷ sevral times, until The [F-ACT] soft key appears.
3. Press the [F-ACT] soft key.
   The fixture offset (ACT) screen displays.

This screen displays the currently selected fixture offset number (P) and fixture offset vector.



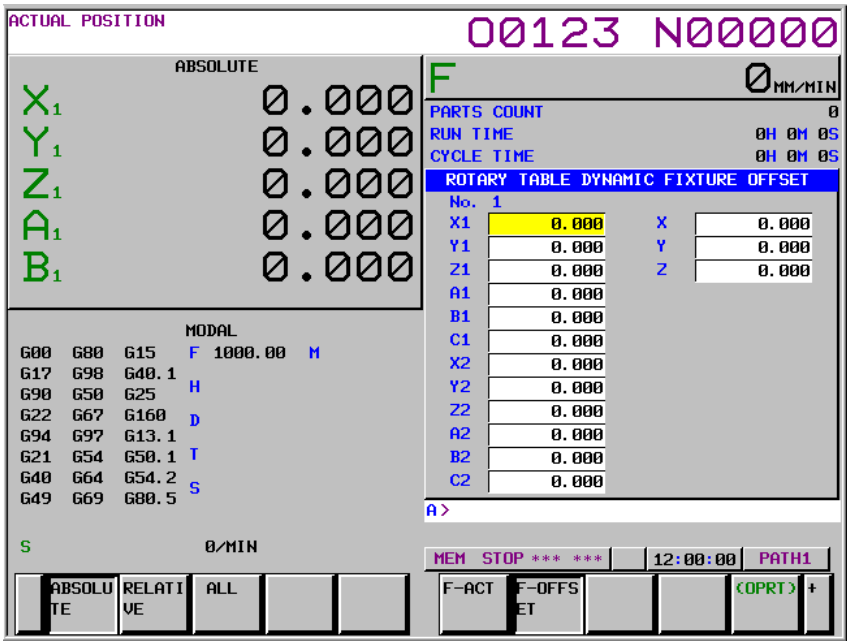
Fixture offset setting screen

Procedure

1 Press function key OFFSET SETTING .

2 Press the continuous menu key sevral times, until The [F-OFFSET] soft key appears.

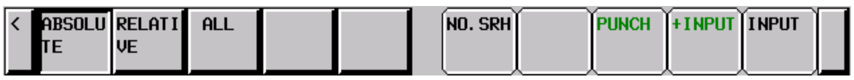
3 Press the [F-OFFSET] soft key.
The fixture offset (ACT) screen displays.
The number of groups that are displayed on one screen is fixed 1 to 4 groups by number of control axis.



Operation

- Entering numeric valuse

- Pres the [OPRT] soft key to displsy the sbove operaion soft key.



- Use the page and cursol keys, and soft key [NO.SRH] to place cursor at a desired items to be set.
- Ente data, then press soft key [INPUT]
- To add a value to already set data, press soft key [+INPU].
  Data can be set using the INPUT MDI key.

- Number of groups of fixture offset values

NO. 01 to NO. 08 indicates the number of a group of fixture offset values.
There are eight groups. Soft key [NO. SRH] can be used to search for a desired group number.

- Reading fixture offset values

Soft key [READ] can be used to output fixture offset values from an external device.

- Outputting fixture offset values

Soft key [PUNCH] can be used to output fixture offset values to an external device.

APPENDIX

A PARAMETERS

This manual describes all parameters indicated in this manual.
For those parameters that are not indicated in this manual and other parameters, refer to the parameter manual.

A.1 DESCRIPTION OF PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
0001							**FCV**	

[Input type] Setting input
[Data type] Bit path

 # 1 FCV Program format
0: Series 16 standard format
1: Series 15 format

1022	**Setting of each axis in the basic coordinate system**

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] 0 to 7

To determine a plane for circular interpolation, cutter compensation, and so forth (G17: Xp-Yp plane, G18: Zp-Xp plane, G19: Yp-Zp plane) and a three-dimensional cutter compensation space (XpYpZp), specify which of the basic three axes (X, Y, and Z) is used for each control axis, or a parallel axis of which basic axis is used for each control axis.
A basic axis (X, Y, or Z) can be specified only for one control axis.
Two or more control axes can be set as parallel axes for the same basic axis.

Setting	Meaning
0	Rotary axis (Neither the basic three axes nor a parallel axis)
1	X axis of the basic three axes
2	Y axis of the basic three axes
3	Z axis of the basic three axes
5	Axis parallel to the X axis
6	Axis parallel to the Y axis
7	Axis parallel to the Z axis

In general, the increment system and diameter/radius specification of an axis set as a parallel axis are to be set in the same way as for the basic three axes.

1411	
	Cutting feedrate

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

[Input type] Setting input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

When the machine requires little change in cutting feedrate during cutting, a cutting feedrate can be specified in the parameter. This eliminates the need to specify a cutting feedrate (F command) in the NC program.

1420	Rapid traverse rate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1430	Maximum cutting feedrate for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

Specify the maximum cutting feedrate for each axis.

1732	Minimum allowable feedrate for the deceleration function based on acceleration in circular interpolation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, degree/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

With the deceleration function based on acceleration in circular interpolation, an optimum feedrate is automatically calculated so that acceleration produced by changing the move direction in circular interpolation does not exceed the maximum allowable acceleration rate specified in parameter No. 1735.
If the radius of an arc is very small, a calculated feedrate may become too low.
In such a case, the feedrate is prevented from decreasing below the value specified in this parameter.

NOTE
During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1735	Maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation for each axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/sec/sec, inch/sec/sec, degree/sec/sec (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (D)
(When the machine system is metric system, 0.0 to +100000.0. When the machine system is inch system, machine, 0.0 to +10000.0.)

Set a maximum allowable acceleration rate for the deceleration function based on acceleration in circular interpolation.
Feedrate is controlled so that acceleration produced by changing the move direction in circular interpolation does not exceed the value specified in this parameter.
For an axis with 0 set in this parameter, the deceleration function based on acceleration is disabled.
If a different value is set in this parameter for each axis, a feedrate is determined from the smaller of the acceleration rates specified for the two circular axes.

> **NOTE**
> During involute interpolation, the minimum allowable feedrate of "clamping of acceleration near a basic circle" in involute interpolation automatic feedrate control is used.

1826	In-position width for each axis

[Input type] Parameter input
[Data type] 2-word axis
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

The in-position width is set for each axis.
When the deviation of the machine position from the specified position (the absolute value of the positioning deviation) is smaller than the in-position width, the machine is assumed to have reached the specified position. (The machine is in the in-position state.)

	#7	#6	#5	#4	#3	#2	#1	#0
3115			APLx					

[Input type] Parameter input
[Data type] Bit axis

 # 5 APLx When the active offset value modification mode based on manual feed is selected, the relative position display is automatically:
0: Not preset.
1: Preset.
Use this parameter when returning a modified offset value to the original value before modification in the active offset value modification mode based on manual feed. The offset value can be returned to the original value by making a movement on the axis by manual feed so that the relative position display (counter) indicates the position 0.

	#7	#6	#5	#4	#3	#2	#1	#0
3290								
							GOF	WOF

[Input type] Parameter input
[Data type] Bit path

 # 0 WOF Setting the tool offset value (tool wear offset) by MDI key input is:
0: Not disabled
1: Disabled (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

> **NOTE**
> When tool offset memory A is selected with the M series, the tool offset set in the parameter WOF is followed even if geometric compensation and wear compensation are not specified with the T series.

 # 1 GOF Setting the tool geometry offset value by MDI key input is:
0: Not disabled
1: Disabled (With parameter No.3294 and No.3295, set the offset number range in which updating the setting is to be disabled.)

3294	Start number of tool offset values whose input by MDI is disabled
3295	Number of tool offset values (from the start number) whose input by MDI is disabled

[Input type] Parameter input
[Data type] Word path
[Valid data range] 0 to 999

When the modification of tool offset values by MDI key input is to be disabled using bit 0 (WOF) of parameter No.3290 and bit 1 (GOF) of parameter No.3290, parameter Nos. 3294 and 3295 are used to set the range where such modification is disabled. In parameter No.3294, set the offset number of the start of tool offset values whose modification is disabled. In parameter No.3295, set the number of such values.

In the following cases, however, none of the tool offset values may be modified:

- When 0 or a negative value is set in parameter No. 3294
- When 0 or a negative value is set in parameter No. 3295
- When a value greater than the maximum tool offset number is set in parameter No. 3294

In the following case, a modification to the values ranging from the value set in parameter No. 3294 to the maximum tool offset number is disabled:

When the value of parameter No. 3294 added to the value of parameter No. 3295 exceeds the maximum tool offset number

When the offset value of a prohibited number is input through the MDI panel, the warning "WRITE PROTECT" is issued.

[Example]

When the following parameter settings are made, modifications to both of the tool geometry offset values and tool wear offset values corresponding to offset numbers 51 to 60 are disabled:

- Bit 1 (GOF) of parameter No. 3290 = 1 (to disable tool geometry offset value modification)
- Bit 0 (WOF) of parameter No. 3290 = 1 (to disable tool wear offset value modification)
- Parameter No. 3294 = 51
- Parameter No. 3295 = 10

If the setting of bit 0 (WOF) of parameter No. 3290 is set to 0 without modifying the other parameter settings above, tool geometry offset value modification only is disabled, and tool wear offset value modification is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
3402								
	G23	CLR			G91	G19	G18	G01

[Input type] Parameter input
[Data type] Bit path

 # 0 G01 G01 Mode entered when the power is turned on or when the control is cleared
0: G00 mode (positioning)
1: G01 mode (linear interpolation)

 # 1 G18 Plane selected when power is turned on or when the control is cleared
0: G17 mode (plane XY)
1: G18 mode (plane ZX)

 # 2 G19 Plane selected when power is turned on or when the control is cleared
0: The setting of bit 1 (G18) of parameter No. 3402 is followed.
1: G19 mode (plane YZ)
When this bit is set to 1, set bit 1 (G18) of parameter No. 3402 to 0.

 # 3 G91 When the power is turned on or when the control is cleared
0: G90 mode (absolute programming)
1: G91 mode (incremental programming)

 # 6 CLR Reset button on the MDI panel, external reset signal, reset and rewind signal, and emergency stop signal
0: Cause reset state.
1: Cause clear state.

 # 7 G23 When the power is turned on
0: G22 mode (stored stroke check on)
1: G23 mode (stored stroke check off)

	#7	#6	#5	#4	#3	#2	#1	#0
3408	C23							

[Input type] Parameter input
[Data type] Bit

C23 If bit 6 (CLR) of parameter No. 3402 is set to 1, set G codes of group number 23 to be placed in the cleared state when the CNC is reset by the reset key of the MDI panel, the external reset signal, the reset & rewind signal, or the emergency stop signal.
The table below indicates the correspondence between bits and G code groups
The setting of a bit has the following meaning:
0: Places the G code group in the cleared state.
1: Does not place G code group in the cleared state.

	#7	#6	#5	#4	#3	#2	#1	#0
5000				ASG				

[Input type] Setting input
[Data type] Bit path

 # 4 ASG When tool compensation memory B/C (M series) or the tool geometry/wear compensation function (T series) is valid, the compensation amount to be modified by the active offset value change mode based on manual feed is:
0: Geometry compensation value
1: Wear compensation value

NOTE
This parameter is valid when the option for tool compensation memory B/C (M series) or tool geometry/wear compensation (T series) is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
5001								
		EVO		EVR	TAL		TLB	TLC

[Input type] Parameter input
[Data type] Bit path

 # 0 TLC
 # 1 TLB These bits are used to select a tool length compensation type.

Type	TLB	TLC
Tool length compensation A	0	0
Tool length compensation B	1	0
Tool length compensation C	-	1

The axis to which cutter compensation is applied varies from type to type as described below.

Tool length compensation A :
Z-axis at all times
Tool length compensation B :
Axis perpendicular to a specified plane (G17/G18/G19)
Tool length compensation C :
Axis specified in a block that specifies G43/G44

 # 3 **TAL** Tool length compensation C
0: Generates an alarm when two or more axes are offset
1: Not generate an alarm even if two or more axes are offset

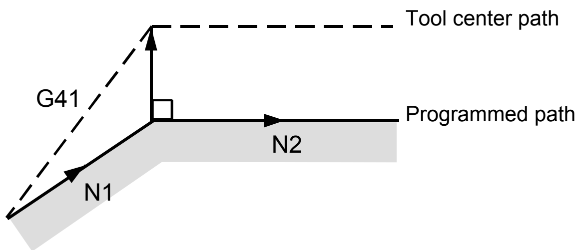
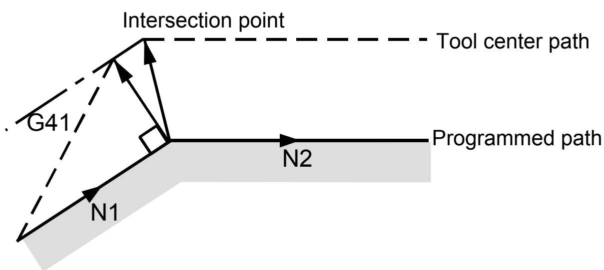
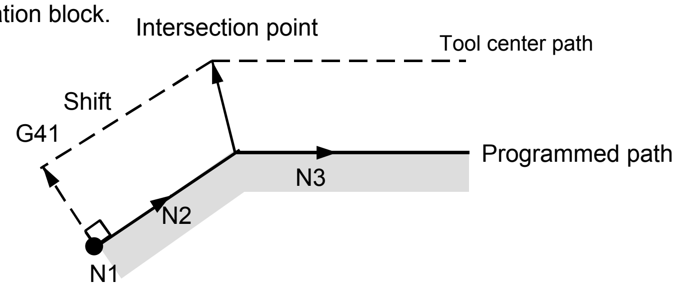
 # 4 **EVR** When a tool compensation value is changed in cutter or tool nose radius compensation mode:
0: Enables the change, starting from that block where the next D or H code is specified.
1: Enables the change, starting from that block where buffering is next performed.

 # 6 **EVO** If a tool compensation value modification is made for tool length compensation A or tool length compensation B in the offset mode (G43 or G44):
0: The new value becomes valid in a block where G43, G44, or an H code is specified next.
1: The new value becomes valid in a block where buffering is performed next.

	#7	#6	#5	#4	#3	#2	#1	#0
5003								
							SUV	SUP

[Input type] Parameter input
[Data type] Bit path

 # 0 SUP
 # 1 SUV These bits are used to specify the type of startup/cancellation of cutter or tool nose radius compensation.

CSC	CSU	Type	Operation
0	0	Type A	A compensation vector perpendicular to the block next to the startup block or the block preceding the cancellation block is output. 
0	1	Type B	A compensation vector perpendicular to the startup block or cancellation block and an intersection vector are output. 
1	0 1	Type C	When the startup block or cancellation block specifies no movement operation, the tool is shifted by the cutter compensation amount in a direction perpendicular to the block next to the startup or the block before cancellation block.  When the block specifies movement operation, the type is set according to the SUP setting; if SUP is 0, type A is set, and if SUP is 1, type B is set.

> **NOTE**
> When SUV,SUP = 0,1 (type B), an operation equivalent to that of Series 16*i*-T is performed.

	#7	#6	#5	#4	#3	#2	#1	#0
5005								
			QNI					

[Input type] Parameter input
[Data type] Bit path

 # 5 QNI With the tool length measurement function, a tool compensation number is selected by:
0: Operation through the MDI panel by the operator (selection based on cursor operation).
1: Signal input from the PMG.

	#7	#6	#5	#4	#3	#2	#1	#0
5006								
		TOS						

[Input type] Parameter input
[Data type] Bit

 # 6 TOS Set a tool length compensation operation.
0: Tool length compensation is performed by an axis movement.
1: Tool length compensation is performed by shifting the coordinate system.

	#7	#6	#5	#4	#3	#2	#1	#0
5007								
				WMH	WMA	TMA	TC3	TC2

[Input type] Parameter input
[Data type] Bit path

 # 0 TC2
 # 1 TC3 If a tool length compensation value is set by pressing the [MEASURE] or [+MEASURE] soft key in tool length measurement, the tool automatically moves to the tool change position. Specify at which reference position the tool change position is located.

TC3	TC2	Meaning
0	0	The tool change position is at the first reference position.
0	1	The tool change position is at the second reference position.
1	0	The tool change position is at the third reference position.
1	1	The tool change position is at the fourth reference position.

 # 2 TMA
0: Tool length measurement is enabled along the Z-axis (axis set to 3 in the parameter No. 1022) only.
1: Tool length measurement is enabled along each axis.

 # 3 WMA

0: Surface-based measurement of a workpiece origin offset value is enabled along the Z-axis only.
1: Surface-based measurement of a workpiece origin offset value is enabled along each axis.

 # 4 WMH

0: Hole-based measurement of a workpiece origin offset value is disabled.
1: Hole-based measurement of a workpiece origin offset value is enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5008					CNV		CNC	

[Input type] Parameter input
[Data type] Bit path

 # 1 CNC
 # 3 CNV These bits are used to select an interference check method in the cutter or tool nose radius compensation mode.

CNV	CNC	Operation
0	0	Interference check is enabled. The direction and the angle of an arc are checked.
0	1	Interference check is enabled. Only the angle of an arc is checked.
1	-	Interference check is disabled.

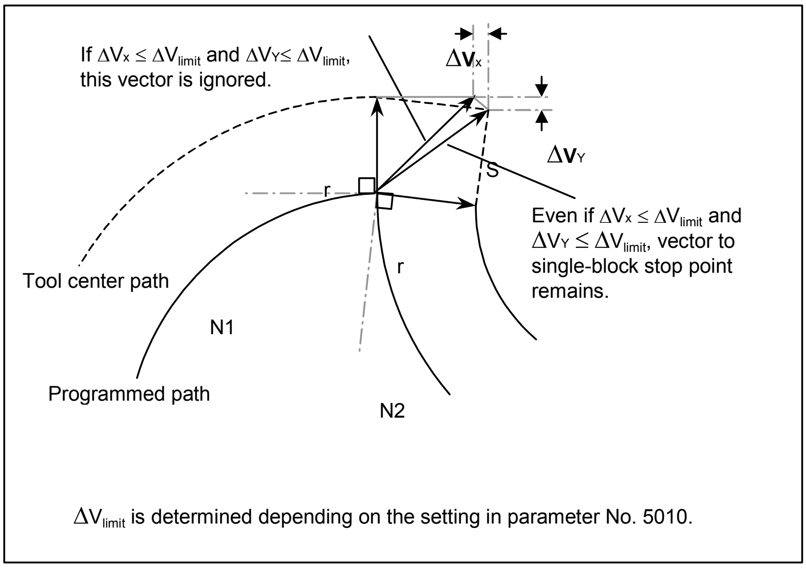
For the operation taken when the interference check shows the occurrence of an interference (overcutting) , see the description of bit 5 (CAV) of parameter No. 19607.

> **NOTE**
> Checking of only the direction cannot be set.

5010	**Limit for ignoring the small movement resulting from cutter or tool nose radius**

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

When the tool moves around a corner in cutter or tool nose radius compensation mode, the limit for ignoring the small travel amount resulting from compensation is set. This limit eliminates the interruption of buffering caused by the small travel amount generated at the corner and any change in feedrate due to the interruption.



5011	Constant denominator for three-dimensional cutter compensation or tool length compensation in a specified direction

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the value of p in the expressions used for finding a three-dimensional cutter compensation vector:

$$Vx = i \times r / p$$
$$Vy = j \times r / p$$
$$Vz = k \times r / p$$

where,

Vx, Vy, Vz : Components of a three-dimensional cutter compensation vector along the X-axis, Y-axis, and Z-axis, or their parallel axes
i, j, k : Values specified in addresses I, J, and K in the program
r : Compensation value
p : Value set in this parameter

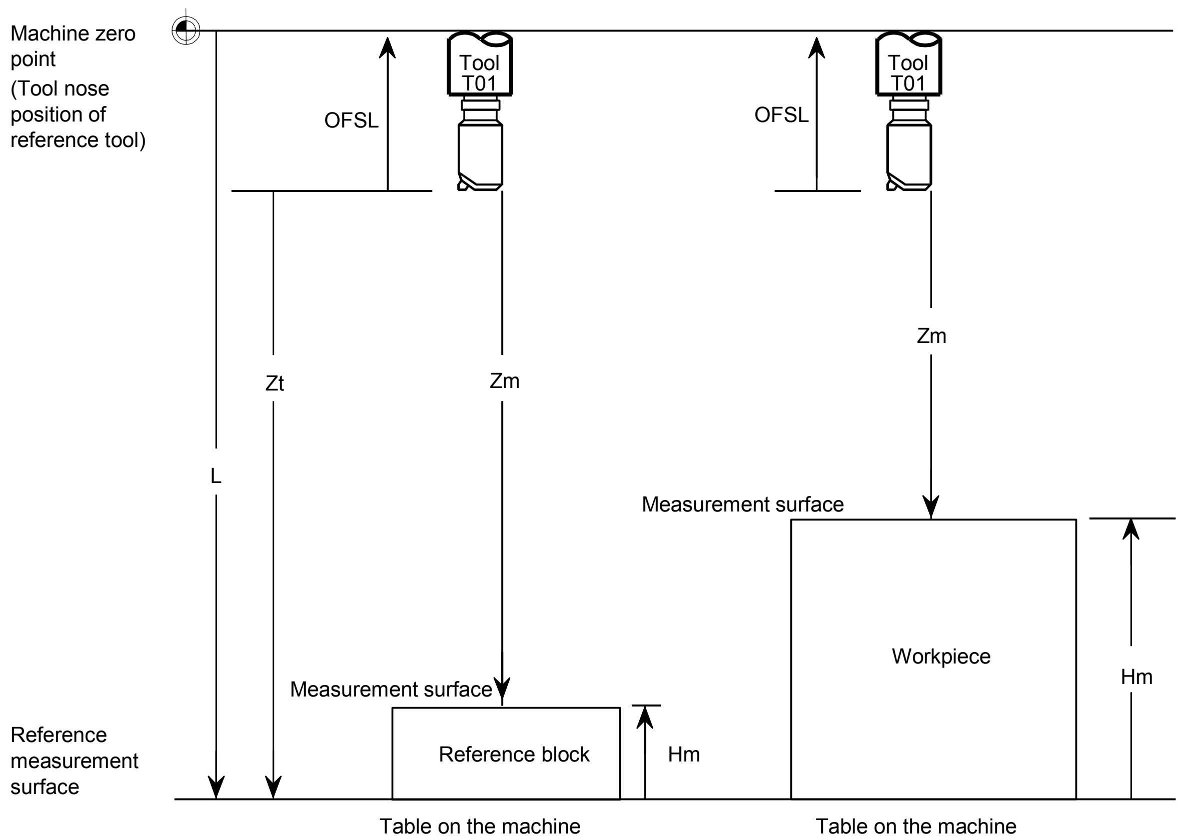
When 0 is set in this parameter, the following is assumed:

$$p = \sqrt{I^2 + J^2 + K^2}$$

5022	Distance (L) from reference tool tip position to the reference measurement surface

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm, inch (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

For each axis, this parameter sets the distance from the reference tool tip position to the reference measurement surface when the machine is at the machine zero point.



L: Distance from the reference tool nose to the reference measurement surface (machine coordinates of the reference measurement surface)
Hm: Distance from the reference measurement surface to actual measurement surface
Zm: Distance from the tool tip of the measured tool at the machine zero point to the measurement surface
Zt: Distance from the tool tip of the measured tool at the machine zero point to the reference measurement surface
OFSL: Tool length compensation (OFSL = Zm-Hm-L)

	#7	#6	#5	#4	#3	#2	#1	#0
5041		AON						

[Input type] Parameter input
[Data type] Bit path

 # 6 AON If a change is made to a tool compensation value (tool length compensation value used with tool length compensation A/B in the case of the M series):

0: In the case of the M series, the change becomes effective starting with the next block specifying G43, G44, or an H code.
In the case of the T series, the change becomes effective starting with the next block specifying a T code.

1: The change becomes effective starting with the next block to be buffered.

> **NOTE**
> 1 This parameter is valid when bit 6 (EVO) of parameter No. 5001 is set to 0.
> 2 The operation of this parameter set to 1 is valid even if a new compensation value is further changed by MDI input or a G10 command before the new compensation value becomes effective.
> 3 The operation of this parameter set to 1 is invalid if a reset operation is performed before a new compensation value becomes effective.

	#7	#6	#5	#4	#3	#2	#1	#0
5042					OFE	OFD	OFC	OFA

[Input type] Parameter input
[Data type] Bit path

> **NOTE**
> When this parameter is set, the power must be turned off before operation is continued.

 # 0 OFA
 # 1 OFC
 # 2 OFD
 # 3 OFE These bits are used to specify the increment system and valid data range of a tool offset value.

For metric input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.01mm	±9999.99mm
0	0	0	0	0.001mm	±9999.999mm
0	0	1	0	0.0001mm	±9999.9999mm
0	1	0	0	0.00001mm	±9999.99999mm
1	0	0	0	0.000001mm	±999.999999mm

For inch input

OFE	OFD	OFC	OFA	Unit	Valid data range
0	0	0	1	0.001inch	±999.999inch
0	0	0	0	0.0001inch	±999.9999inch
0	0	1	0	0.00001inch	±999.99999inch
0	1	0	0	0.000001inch	±999.999999inch
1	0	0	0	0.0000001inch	±99.9999999inch

	#7	#6	#5	#4	#3	#2	#1	#0
5101								
								FXY

[Input type] Parameter input
[Data type] Bit path

 # 0 FXY The drilling axis in the drilling canned cycle is:
0: Always the Z-axis
1: The axis selected by the program

> **NOTE**
> In the case of the T series, this parameter is valid only for the drilling canned cycle in the Series 15 format.

	#7	#6	#5	#4	#3	#2	#1	#0
5105								
								SBC

[Input type] Parameter input
[Data type] Bit path

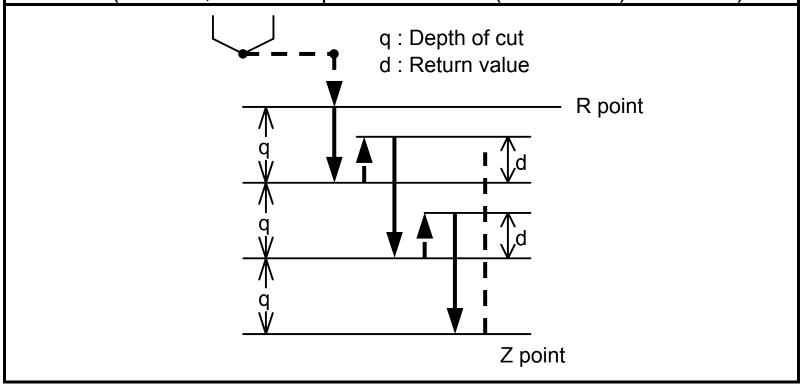
 # 0 SBC In a drilling canned cycle, chamfering, or corner R cycle:
0: A single block stop is not performed.
1: A single block stop is performed.

5114	Return value of high-speed peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets the return value in high-speed peck drilling cycle.

G73 (M series)
G83 (T series, when the parameter RTR (No.5101#2) is set to 0)

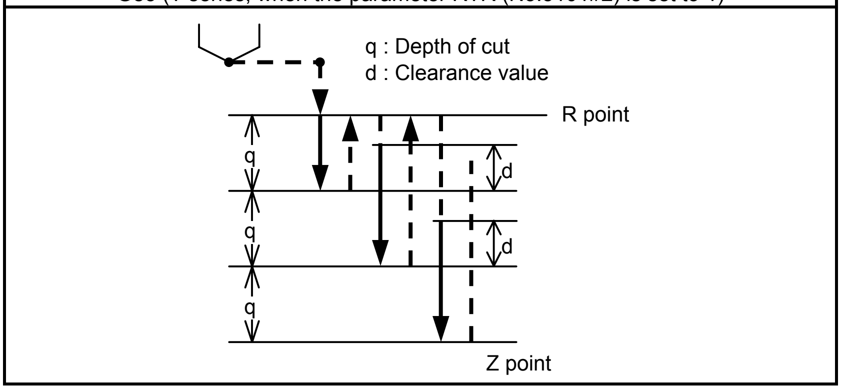


5115	Clearance value in a peck drilling cycle

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

This parameter sets a clearance value in a peck drilling cycle.

G83 (M series)
G83 (T series, when the parameter RTR (No.5101#2) is set to 1)



5148	Tool retraction direction after orientation in a fine boring cycle or back boring cycle

[Input type] Parameter input
[Data type] Byte axis
[Valid data range] -10 to 10

This parameter sets an axis and direction for tool retraction after spindle orientation in a fine boring cycle or back boring cycle. For each boring axis, an axis and direction for tool retraction after orientation can be set. Set a signed axis number.

Example)
Suppose that:
When the boring axis is the X-axis, the tool retraction direction after orientation is -Y.
When the boring axis is the Y-axis, the tool retraction direction after orientation is +Z.
When the boring axis is the Z-axis, the tool retraction direction after orientation is -X.
Then, set the following (assuming that the first, second, and third axes are the X-axis, Y-axis, and Z-axis, respectively):
Set -2 in the parameter for the first axis. (The tool retraction direction is -Y.)
Set 3 in the parameter for the second axis. (The tool retraction direction is -Y.)
Set -1 in the parameter for the third axis. (The tool retraction direction is -X.)
Set 0 for other axes.

	#7	#6	#5	#4	#3	#2	#1	#0
5160						NOL	OLS	

[Input type] Parameter input
[Data type] Bit path

#1 OLS When an overload torque detection signal is received in a peck drilling cycle of a small diameter, the feedrate and spindle speed are:
0: Not changed.
1: Changed.

#2 NOL When the depth of cut per action is satisfied although no overload torque detection signal is received in a peck drilling cycle of a small diameter, the feedrate and spindle speed are:
0: Not changed.
1: Changed.

5163	M code that specifies the peck drilling cycle mode of a small diameter

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 1 to 99999999

This parameter sets an M code that specifies the peck drilling cycle mode of a small diameter.

5164	Percentage of the spindle speed to be changed at the start of the next advancing after an overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted because the overload torque detection signal is received.

$S2 = S1 \times d1 \div 100$

S1: Spindle speed to be changed
S2: Spindle speed changed

Set d1 as a percentage.

> **NOTE**
> When 0 is set, the spindle speed is not changed.

5165	Percentage of the spindle speed to be changed at the start of the next advancing when no overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the spindle speed to be changed at the start of the next advancing after the tool is retracted without the overload torque detection signal received.

$S2 = S1 \times d2 \div 100$

S1: Spindle speed to be changed
S2: Spindle speed changed

Set d2 as a percentage.

> **NOTE**
> When 0 is set, the spindle speed is not changed.

5166	
	Percentage of the cutting feedrate to be changed at the start of the next cutting after an overload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances because the overload torque detection signal is received.

$F2 = F1 \times b1 \div 100$

F1: Cutting feedrate to be changed
F2: Cutting feedrate changed

Set b1 as a percentage.

NOTE
When 0 is set, the cutting feedrate is not changed.

5167	
	Percentage of the cutting feedrate to be changed at the start of the next cutting when no ovarload torque detection signal is received

[Input type] Parameter input
[Data type] Word path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the percentage of the cutting feedrate to be changed at the start of cutting after the tool is retracted and advances without the overload torque detection signal received.

$F2 = F1 \times b2 \div 100$

F1: Cutting feedrate to be changed
F2: Cutting feedrate changed

Set b2 as a percentage.

NOTE
When 0 is set, the cutting feedrate is not changed.

5168	Lower limit of the percentage of the cutting feedrate in a peck drilling cycle of a small diameter

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 1 to 255

This parameter sets the lower limit of the percentage of the cutting feedrate changed repeatedly to the specified cutting feedrate.

$FL = F \times b3 \div 100$

F: Specified cutting feedrate

FL: Changed cutting feedrate

Set b3 as a percentage.

5170	Number of the macro variable to which to output the total number of retractions during cutting

[Input type] Parameter input
[Data type] Word path
[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted during cutting. The total number cannot be output to common variables #500 to #599.

5171	Number of the macro variable to which to output the total number of retractions because of the reception of an overload torque detection signal

[Input type] Parameter input
[Data type] Word path
[Valid data range] 100 to 149

This parameter sets the number of the custom macro common variable to which to output the total number of times the tool is retracted after the overload torque detection signal is received during cutting. The total number cannot be output to common variables #500 to #599.

5172	
	Feedrate of retraction to point R when no address I is specified

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
This parameter sets the feedrate of retraction to point R when no address I is specified.

5173	
	Feedrate of advancing to the position just before the bottom of a hole when no address I is specified

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
This parameter sets the feedrate of advancing to the position just before the bottom of a previously machined hole when no address I is specified.

5174	
	Clearance in a peck drilling cycle of a small diameter

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min (machine unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
This parameter sets the clearance in a peck drilling cycle of a small diameter.

	#7	#6	#5	#4	#3	#2	#1	#0
5200								
		FHD	PCP	DOV				G84

[Input type] Parameter input
[Data type] Bit path

 # 0 G84 Method for specifying rigid tapping
0: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
1: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

 # 4 DOV Override during extraction in rigid tapping
0: Invalidated
1: Validated (The override value is set in parameter No.5211. However, set an override value for rigid tapping return in parameter No. 5381.)

 # 5 PCP Rigid tapping
0: Used as a high-speed peck tapping cycle
1: Not used as a high-speed peck tapping cycle

 # 6 FHD Feed hold and single block in rigid tapping
0: Invalidated
1: Validated

	#7	#6	#5	#4	#3	#2	#1	#0
5201				OV3	OVU			

[Input type] Parameter input
[Data type] Bit path

 # 3 OVU The increment unit of the override parameter (No.5211) for tool rigid tapping extraction is:
0: 1%
1: 10%

 # 4 OV3 A spindle speed for extraction is programmed, so override for extraction operation is:
0: Disabled.
1: Enabled.

	#7	#6	#5	#4	#3	#2	#1	#0
5203				OVS				

[Input type] Parameter input
[Data type] Bit path

 # 4 OVS In rigid tapping, override by the feedrate override select signal and cancellation of override by the override cancel signal is:
0: Disabled.
1: Enabled.
When feedrate override is enabled, extraction override is disabled.
The spindle override is clamped to 100% during rigid tapping, regardless of the setting of this parameter.

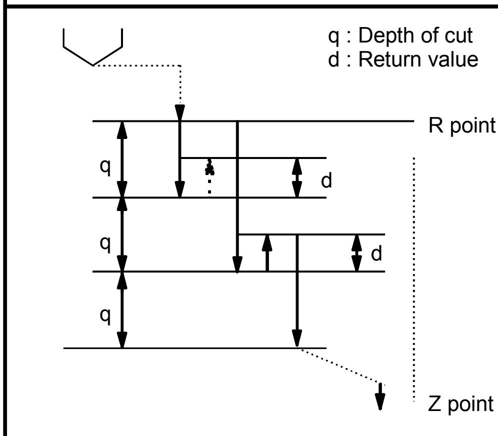
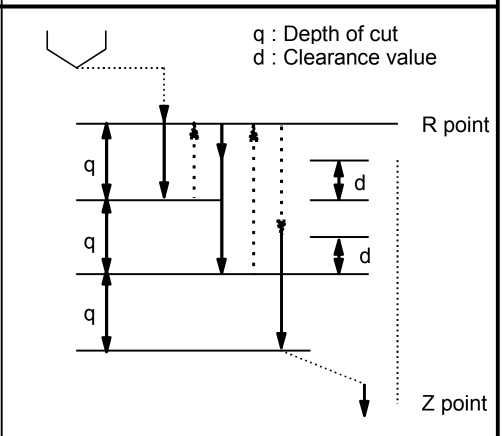
5211	Override value during rigid tapping extraction

[Input type] Parameter input
[Data type] Word path
[Unit of data] 1% or 10%
[Valid data range] 0 to 200
The parameter sets the override value during rigid tapping extraction.

NOTE
The override value is valid when DOV in parameter No.5200 #4 is "1". When OVU (bit 3 of parameter No.5201) is 1, the unit of set data is 10%. An override of up to 200% can be applied to extraction.

5213	Return in peck rigid tapping cycle

[Input type] Setting input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the drilling axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)
This parameter sets the return or clearance in the peck tapping cycle.

When the parameter PCP (bit 5 of No.5200) is set to 0.	When the parameter PCP (bit 5 of No.5200) is set to 1.
	

5241	Maximum spindle speed in rigid tapping (first gear)
5242	Maximum spindle speed in rigid tapping (second gear)
5243	Maximum spindle speed in rigid tapping (third gear)
5244	Maximum spindle speed in rigid tapping (fourth gear)

[Input type] Parameter input
[Data type] 2-word spindle
[Unit of data] min^{-1}
[Valid data range] 0 to 9999

Spindle position coder gear ratio
1 : 1 0 to 7400
1 : 2 0 to 9999
1 : 4 0 to 9999
1 : 8 0 to 9999

Each of these parameters is used to set a maximum spindle speed for each gear in rigid tapping.
Set the same value for both parameter No.5241 and parameter No.5243 for a one-stage gear system. For a two-stage gear system, set the same value as set in parameter No. 5242 in parameter No. 5243. Otherwise, alarm PS0200 will be issued. This applies to the M series.

5321	Spindle backlash in rigid tapping (first-stage gear)
5322	Spindle backlash in rigid tapping (second-stage gear)
5323	Spindle backlash in rigid tapping (third-stage gear)
5324	Spindle backlash in rigid tapping (fourth-stage gear)

[Input type] Parameter input
[Data type] Word spindle
[Unit of data] Detection unit
[Valid data range] -9999 to 9999

Each of these parameters is used to set a spindle backlash.

	#7	#6	#5	#4	#3	#2	#1	#0
5400								RIN

[Input type] Parameter input
[Data type] Bit path

 # 0 RIN Coordinate system rotation angle command (R)
0: Specified by an absolute programming
1: Specified by an absolute programming (G90) or incremental programming (G91)

5410	Angular displacement used when no angular displacement is specified for coordinate system rotation

[Input type] Setting input
[Data type] 2-word path
[Unit of data] 0.001 degree
[Valid data range] -360000 to 360000

Set an angular displacement for coordinate system rotation. When no angular displacement is specified with address R for coordinate system rotation in a block specifying G68, the setting of this parameter is used as the angular displacement for coordinate system rotation.

	#7	#6	#5	#4	#3	#2	#1	#0
5431								MDL

[Input type] Parameter input
[Data type] Bit path

NOTE
When this parameter is set, the power must be turned off before operation is continued.

 # 0 MDL The G60 code (single direction positioning) is:
0: One-shot G code (group 00).
1: Modal G code (group 01).

5480	Number of the axis for controlling the normal direction

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to the maximum controlled axis number

This parameter sets the controlled axis number of the axis for controlling the normal direction.

5481	
	Feedrate of rotation of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real axis
[Unit of data] deg/min
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)

This parameter sets the feedrate of the movement along the normal direction controlled axis that is inserted at the start point of a block during normal direction control.

5482	
	Limit value used to determine whether to ignore the rotation insertion of the normal direction controlled axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))

The rotation block of the normal direction controlled axis is not inserted when the rotation insertion angle calculated during normal direction control does not exceed this setting.
The ignored rotation angle is added to the next rotation insertion angle, and the block insertion is then judged.

NOTE
1 No rotation block is inserted when 360 or more degrees are set.
2 If 180 or more degrees are set, a rotation block is inserted only when the circular interpolation setting is 180 or more degrees.

	#7	#6	#5	#4	#3	#2	#1	#0
5500				G90	INC	ABS	REL	

[Input type] Parameter input
[Data type] Bit path

 # 1 REL The indication of the position of the index table indexing axis in the relative coordinate system is:
0: Not rounded to one rotation.
1: Rounded to one rotation.

 # 2 ABS The indication of the position of the index table indexing axis in the absolute coordinate system is:
0: Not rounded to one rotation.
1: Rounded to one rotation.

 # 3 **INC** If the M code for specifying negative direction rotation (parameter No. No. 5511) is not set, the shortcut rotation direction in the G90 mode is:
0: Not used.
1: Used.

 # 4 **G90** The command for the index table indexing axis:
0: Follows the absolute/incremental programming.
1: Is regarded as an absolute programming at all times.

5511	M code specifying negative direction rotation for index table indexing

[Input type] Parameter input
[Data type] 2-word path
[Valid data range] 0 to 99999999
0: The move direction of the index table indexing axis is determined by bit 3 (INC) of parameter No. 5500 and the command.
1 to 99999999: The index table indexing axis moves in the positive direction at all times. The index table indexing axis moves in the negative direction only when an M code set with a move command is specified.

NOTE
Be sure to set bit 2 (ABS) of parameter No. 5500 to 1.

5512	Minimum positioning angle for the index table indexing axis

[Input type] Parameter input
[Data type] Real path
[Unit of data] Degree
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
Set a minimum positioning angle (angular displacement) for the index table indexing axis. An angular displacement in a positioning command must be an integral multiple of the setting of this parameter. When 0 is set, no angular displacement is checked.
Not only commands but also coordinate system settings and workpiece origin offset values are checked for a minimum positioning angle.

5610	Limit of initial permissible error during involute interpolation

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 0 or positive 9 digit of minimum unit of data (refer to the standard parameter setting table (B))
(When the increment system is IS-B, 0.0 to +999999.999)

This parameter sets the allowable limit of deviation between an involute curve passing through a start point and an involute curve passing through an end point for an involute interpolation command.

5620	Lower override limit in automatic feedrate control during involute interpolation

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100

In "override in the cutter compensation mode" under involute interpolation automatic feedrate control, the feedrate of the tool center near a basic circle may become very low in the case of an inner offset. To prevent this, set a lower override limit in this parameter.
Thus, the feedrate is clamped so that the feedrate is not lower than a specified feedrate multiplied by the lower override limit set in this parameter.

NOTE
When 0 or a value not within the valid data range is set, involute interpolation automatic feedrate control ("override in the cutter compensation mode" and "acceleration clamping near a basic circle") is disabled.

	#7	#6	#5	#4	#3	#2	#1	#0
6210		MDC						

[Input type] Parameter input
[Data type] Bit path

 # 6 MDC The measurement result of automatic tool length measurement (M series) or automatic tool compensation (T series) is:
0: Added to the current offset.
1: Subtracted from the current offset.

6241	**Feedrate during measurement of automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)**

6242	**Feedrate during measurement of automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)**

6243	**Feedrate during measurement of automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)**

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

These parameters set the relevant feedrate during measurement of automatic tool compensation (T series) or automatic tool length measurement (M series).

NOTE
When the setting of parameter No. 6242 or 6243 is 0, the setting of parameter No. 6241 is used.

6251	**γ value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)**

6252	**γ value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)**

6253	**γ value during automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)**

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the relevant γ value during automatic tool compensation (T series) or automatic tool length measurement (M series).

> **NOTE**
> 1 For the M series, when the setting of parameter No. 6252 or 6253 is 0, the setting of parameter No. 6251 is used.
> 2 Set a radius value regardless of whether diameter or radius programming is specified.

6254	
	ε value during automatic tool length measurement (M series) (for the XAE1 and GAE1 signals)

6255	
	ε value during automatic tool length measurement (M series) (for the XAE2 and GAE2 signals)

6256	
	ε value during automatic tool length measurement (M series) (for the XAE3 and GAE3 signals)

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] mm, inch, deg (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)

These parameters set the relevant ε value during automatic tool compensation (T series) or automatic tool length measurement (M series).

> **NOTE**
> 1 For the M series, when the setting of parameter No. 6255 or 6256 is 0, the setting of parameter No. 6254 is used.
> 2 Set a radius value regardless of whether diameter or radius programming is specified.

	#7	#6	#5	#4	#3	#2	#1	#0
7570								
					CFA			FTP

[Input type] Parameter input
[Data type] Bit path

 # 0 FTP Fixture offset type setting
0: Movement type
(The tool moves when the fixture offset changes.)
1: Shift type
(The tool does not move when the fixture offset changes.)

 # 3 CFA When the fixture offset function is used, and a rotation axis is specified in the increment mode (G91 mode) after manual intervention in the state where the manual absolute switch is on:
0: A vector calculation is made using coordinates not reflecting a manual intervention amount.
1: A vector calculation is made using coordinates reflecting a manual intervention amount.

	#7	#6	#5	#4	#3	#2	#1	#0
7575								
								FAX

[Input type] Parameter input
[Data type] Bit axis

 # 0 FAX Fixture offset on each axis is:
0: Disabled.
1: Enabled.

7580	Rotation axis for fixture offset (first group)

7581	Linear axis 1 for fixture offset (first group)

7582	Linear axis 2 for fixture offset (first group)

7583	Rotation axis for fixture offset (second group)

7584	Linear axis 1 for fixture offset (second group)

7585	Linear axis 2 for fixture offset (second group)

7586	Rotation axis for fixture offset (third group)

7587	Linear axis 1 for fixture offset (third group)

7588	Linear axis 2 for fixture offset (third group)

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 0 to Number of controlled axes

These parameters specify rotation axes for fixture offset and pairs of linear axes for selecting a rotation plane. Specify a pair of linear axes so that rotation from the positive direction of linear axis 1 to the positive direction is in the normal direction of the rotation axis.

Up to three groups of a rotation axis setting and two linear axis settings can be specified. The fixture offset value is calculated first for the rotation axis in the first group. Then, for the second and third groups, the fixture value is sequentially calculated using the previous calculation result. When you do not need the third group, set 0 for the rotation axis.

	#7	#6	#5	#4	#3	#2	#1	#0
8360								ROV

[Input type] Parameter input
[Data type] Bit path

 # 0 ROV As rapid traverse override for a section from the chopping start point to point R:
0: Chopping override is used.
1: Rapid traverse override is used.

8370	Chopping axis

[Input type] Parameter input
[Data type] Byte path
[Valid data range] 1 to Number of controlled axes

This parameter sets which servo axis the chopping axis corresponds to.

8371	Chopping reference point (point R)

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8372	Chopping upper dead point

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8373	Chopping lower dead point

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm, inch, deg (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] 9 digit of minimum unit of data (refer to standard parameter setting table (A))
(When the increment system is IS-B, -999999.999 to +999999.999)
The data set in this parameter is absolute coordinates.

8374	Chopping feedrate

[Input type] Parameter input
[Data type] Real path
[Unit of data] mm/min, inch/min, deg/min (input unit)
[Minimum unit of data] Depend on the increment system of the reference axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)
This parameter sets the chopping feedrate.

8375	Maximum chopping feedrate

[Input type] Parameter input
[Data type] Real axis
[Unit of data] mm/min, inch/min, deg/min (machine unit)
[Minimum unit of data] Depend on the increment system of the applied axis
[Valid data range] Refer to the standard parameter setting table (C)
(When the increment system is IS-B, 0.0 to +240000.0)

The chopping feedrate is clamped to the setting of this parameter. When this parameter is set to 0 for the chopping axis, the chopping feedrate is clamped to the rapid traverse rate (parameter No. 1420).

8376	Chopping compensation factor

[Input type] Parameter input
[Data type] Byte path
[Unit of data] %
[Valid data range] 0 to 100

The value obtained by multiply the sum of the servo delay in an chopping operation and the acceleration/deceleration delay by the rate set in this parameter is used as chopping delay compensation. When this parameter is set to 0, chopping delay compensation is not applied.

8377	Chopping compensation start tolerance

[Input type] Parameter input
[Data type] 2-word path
[Unit of data] Detection unit
[Valid data range] 0 to 99999999

In a chopping operation, compensation is applied when the difference between an amount of shortage at the upper dead point and that at the lower dead point due to the servo position control delay is less than the value set in this parameter. When this parameter is set to 0, compensation is not applied.

	#7	#6	#5	#4	#3	#2	#1	#0
19607								
		NAA	CAV			CCC		

[Input type] Parameter input
[Data type] Bit path

 # 2 CCC In the cutter or tool nose radius compensation mode, the outer corner connection method is based on:
0: Linear connection type.
1: Circular connection type.

 # 5 **CAV** When an interference check finds that interference (overcutting) occurred:

0: Machining stops with the alarm (PS0041).
(Interference check alarm function)

1: Machining is continued by changing the tool path to prevent interference (overcutting) from occurring. (Interference check avoidance function)

For the interference check method, see the descriptions of bit 1 (CNC) of parameter No. 5008 and bit 3 (CNV) of parameter No. 5008.

 # 6 **NAA** When the interference check avoidance function considers that an avoidance operation is dangerous or that a further interference to the interference avoidance vector occurs:

0: An alarm is issued.
When an avoidance operation is considered to be dangerous, the alarm (PS5447) is issued.
When a further interference to the interference avoidance vector is considered to occur, the alarm (PS5448) is issued.

1: No alarm is issued, and the avoidance operation is continued.

> **NOTE**
> Usually, set this parameter to 0.

19625	Number of blocks to be read in the cutter or tool nose radius compensation mode

[Input type] Setting input
[Data type] Byte path
[Valid data range] 3 to 8

This parameter sets the number of blocks to be read in the cutter or tool nose radius compensation mode. When a value less than 3 is set, the specification of 3 is assumed. When a value greater than 8 is set, the specification of 8 is assumed. As a greater number of blocks are read, an overcutting (interference) forecast can be made with a command farther ahead. However, the number of blocks read and analyzed increases, so that a longer block processing time becomes necessary.

Even if the setting of this parameter is modified in the MDI mode by stopping in the cutter or tool nose radius compensation mode, the setting does not become valid immediately. Before the new setting of this parameter can become valid, the cutter or tool noise radius compensation mode must be canceled, then the mode must be entered again.

A.2 DATA TYPE

Parameters are classified by data type as follows:

Data type	Valid data range	Remarks
Bit Bit machine group Bit path Bit axis Bit spindle	0 or 1	
Byte Byte machine group Byte path Byte axis Byte spindle	-128 to 127 0 to 255	Some parameters handle these types of data as unsigned data.
Word Word machine group Word path Word axis Word spindle	-32768 to 32767 0 to 65535	Some parameters handle these types of data as unsigned data.
2-word 2-word machine group 2-word path 2-word axis 2-word spindle	0 to ±999999999	Some parameters handle these types of data as unsigned data.
Real Real machine group Real path Real axis Real spindle	See the Standard Parameter Setting Tables.	

NOTE

1. Each of the parameters of the bit, bit machine group, bit path, bit axis, and bit spindle types consists of 8 bits for one data number (parameters with eight different meanings).
2. For machine group types, parameters corresponding to the maximum number of machine groups are present, so that independent data can be set for each machine group.
3. For path types, parameters corresponding to the maximum number of paths are present, so that independent data can be set for each path.
4. For axis types, parameters corresponding to the maximum number of control axes are present, so that independent data can be set for each control axis.
5. For spindle types, parameters corresponding to the maximum number of spindles are present, so that independent data can be set for each spindle axis.
6. The valid data range for each data type indicates a general range. The range varies according to the parameters. For the valid data range of a specific parameter, see the explanation of the parameter.

A.3 STANDARD PARAMETER SETTING TABLES

This section defines the standard minimum data units and valid data ranges of the CNC parameters of the real type, real machine group type, real path type, real axis type, and real spindle type. The data type and unit of data of each parameter conform to the specifications of each function.

NOTE
1. Values are rounded up or down to the nearest multiples of the minimum data unit.
2. A valid data range means data input limits, and may differ from values representing actual performance.
3. For information on the ranges of commands to the CNC, refer to Appendix D, "Range of Command Value."

(A) Length and angle parameters (type 1)

Unit of data	Increment	Minimum	Valid data range
mm deg.	IS-A	0.01	-999999.99 to +999999.99
	IS-B	0.001	-999999.999 to +999999.999
	IS-C	0.0001	-99999.9999 to +99999.9999
	IS-D	0.00001	-9999.99999 to +9999.99999
	IS-E	0.000001	-999.999999 to +999.999999
inch	IS-A	0.001	-99999.999 to +99999.999
	IS-B	0.0001	-99999.9999 to +99999.9999
	IS-C	0.00001	-9999.99999 to +9999.99999
	IS-D	0.000001	-999.999999 to +999.999999
	IS-E	0.0000001	-99.9999999 to +99.9999999

(B) Length and angle parameters (type 2)

Unit of data	Increment system	Minimum data unit	Valid data range
mm deg.	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

(C) Velocity and angular velocity parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/min degree/min	IS-A	0.01	0.00 to +999000.00
	IS-B	0.001	0.000 to +999000.000
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch/min	IS-A	0.001	0.000 to +96000.000
	IS-B	0.0001	0.0000 to +9600.0000
	IS-C	0.00001	0.00000 to +4000.00000
	IS-D	0.000001	0.000000 to +400.000000
	IS-E	0.0000001	0.0000000 to +40.0000000

(D)Acceleration and angular acceleration parameters

Unit of data	Increment system	Minimum data unit	Valid data range
mm/sec^2 deg./sec^2	IS-A	0.01	0.00 to +999999.99
	IS-B	0.001	0.000 to +999999.999
	IS-C	0.0001	0.0000 to +99999.9999
	IS-D	0.00001	0.00000 to +9999.99999
	IS-E	0.000001	0.000000 to +999.999999
inch/sec^2	IS-A	0.001	0.000 to +99999.999
	IS-B	0.0001	0.0000 to +99999.9999
	IS-C	0.00001	0.00000 to +9999.99999
	IS-D	0.000001	0.000000 to +999.999999
	IS-E	0.0000001	0.0000000 to +99.9999999

INDEX

Revision Record

FANUC Series 30*i*/300*i*/300*i*s-MODEL A, Series 31*i*/310*i*/310*i*s-MODEL A5, Series 31*i*/310*i*/310*i*s-MODEL A, Series 32*i*/320*i*/320*i*s-MODEL A USER'S MANUAL (For Machining Center System)(B-63944EN-2)

Edition	Date	Contents	Edition	Date	Contents
02	Jun, 2004	Addition of functions Addition of following models - Series 31*i* /310*i* /310*i*s-MODEL A5 - Series 31*i* /310*i* /310*i*s-MODEL A - Series 32*i* /320*i* /320*i*s-MODEL A			
01	Jul., 2003				